USAID | JORDAN
FROM THE AMERICAN PEOPLE

WATER REUSE AND ENVIRONMENTAL CONSERVATION PROJECT

CONTRACT NO. EDH–I–00–08–00024–00 ORDER NO. 04

WaterGEMS Training Manual
November 2014

IMPLEMENTED BY AECOM

November 2014

This document was produced for review by the United States Agency for International Development. It was prepared by AECOM.

WATER REUSE AND ENVIRONMENTAL
CONSERVATION PROJECT
CONTRACT NO. EDH–I–00–08–00024–00 ORDER NO. 04

WATERGEMS TRAINING MANUAL

NOVEMBER 2014

Submitted to:
USAID Jordan

Prepared by:
AECOM

Table of Contents

LIST OF ACRONYMS AND DEFINITION OF TERMS

A

Age
An analysis for the age of water determines how long the water has been in the system, and is a general water quality indicator.

ArcObjects
ArcObjects is the framework upon which ArcGIS has been built. It is a collection of software components based on the COM protocol, which allows for the customization and extension of the core software functionality.

Available Fire Flow
Amount of flow available at a node for fire protection while maintaining all fire flow pressure constraints.

B

.bak
Extension for backup files.

Base Elevation & Level
Elevation from which all tank levels are measured. For example, a tank level of two meters represents a water surface elevation two meters above the base elevation.

Boundary Node
Node with a known hydraulic grade. It may be static (unchanging with time), such as a reservoir, or dynamic (changes with time), such as a tank. Every pipe network must contain at least one boundary node. In order to compute the hydraulic grade at the other nodes in the network, they must be reachable from a boundary.

Bulk Reaction Coefficient
Coefficient used to define how rapidly a constituent grows or decays over time. It is expressed in units of 1/time, for first-order reactions.

C

Calc. Min. System Pressure
Minimum calculated pressure of all junctions in the system during fire flow withdrawal at a node.

Calc. Min. Zone Pressure
Minimum calculated pressure of all junctions in the same zone as the node where fire flow withdrawal occurs.

Calc. Residual Pressure
Calculated pressure at the junction node where the fire flow withdrawal occurs.

Calculation Unready
An element that does not have all the required information for performing an analysis is considered to be calculation unready.

C-Coefficient
Roughness coefficient used in the Hazen-Williams Equation.

Check Valve
Prevents water from flowing backwards through the pipe. In other words, water can only flow from the “From” Node to the “To” Node.

Closed/Inactive Status
You can control the status of a valve to be either inactive or closed. Inactive means that the valve will act like an open pipe where flow can occur in either direction, and the headloss across the valve will be calculated using the valve's minor loss factor. Closed means that no flow will occur through the valve.

Constituent
Any substance, such as chlorine or fluoride, for which the growth or decay can be adequately described through the use of a bulk reaction coefficient and a wall reaction coefficient.

Context Menu
A shortcut menu opened by right-clicking a project element or data entry field. Commands on the context menu are specific to the current state of the selected item.

Control Status
A pressure pipe can be either Open or Closed. Open means that flow occurs in the pipe, and Closed means that no flow occurs in the pipe.

Conveyance Element
A pipe or channel used to transport water.

Coordinates
Distances perpendicular to a set of reference axes. Some areas may have predefined coordinate systems, while other coordinate systems may be arbitrary. Coordinates may be presented as X and Y values or may be defined as Northing and Easting values, depending on individual preferences.

Coverage
A collection of data that has a common theme, and is considered a single unit.

Cross Section Type
Tanks can have either a constant area cross section or a variable area cross section. The cross section of a tank with a constant area is the same throughout the depth. The cross section of a tank with a variable area varies throughout the depth.

Crosshair
The cursor that looks like a plus sign (+).

Current Storage Volume
The volume of water currently stored in a tank. It includes both the hydraulically active volume and the hydraulically inactive volume.

CV
Check valve.

D

.dgn:
Drawing information in MicroStation.

.**dwg:**
Drawing information in AutoCAD.

.dwh
Drawing information in Stand-Alone.

Database Connections
A connection represented by a group of database links. There may be a single linked external file within a connection, or there may be several external file links within a single connection.

Dataset
A Bentley WaterGEMS V8*i* project.

DBMS
An acronym that stands for Database Management System. These systems can be relational (RDBMS) or non-relational.

DEM
Digital elevation model.

Demand
Represents the total demand from an individual junction for the current time period. It is based on the information from the Demand tab of the Junction Editor.

Design Point
Point at which a pump was originally intended to operate, and is typically the best efficiency point (BEP) of the pump. At discharges above or below this point, the pump is not operating under optimum conditions.

Diameter
Refers to a pipe or valve's inside diameter. It is the distance between two internal points directly opposite each other.

Discharge
Volumetric rate of flow given in units of length^3/time.

DLG
Digital line graph.

Double-Click
To click the left mouse button twice in rapid succession.

Drag
To hold down one of the mouse buttons while you move the mouse.

E

Element
An object such as a tank, junction node, or pipe in a drawing.

Elevation
The distance from a datum plane to the center of the element. Elevations are often referenced with mean sea level as the datum elevation.

Energy Grade Line (EGL)
Sum of datum (base elevation), elevation, velocity head, and pressure head at a section.

EPS
Extended Period Simulation.

Extended Edit
A small button with an ellipsis (...) as the label. Extended edit buttons are located next to drop-down choice lists and provide further editing for the associated choice list items.

External Files
Any files outside of this program that can be linked. These include database files (such as Dbase) and spreadsheets (such as Excel). Throughout the documentation, all of these file types will be referred to as databases or external files interchangeably.

Extrapolate
To infer a value based on other known values, with the desired value lying outside the known range. Often based upon extending the slope of the line connecting the previous known values to the desired point. See also “interpolate.”

F

Feature Class
1. A classification describing the format of geographic features and supporting data in a coverage. Coverage feature classes for representing geographic features include point, arc, node, route-system, route, section, polygon and region. One or more coverage features are used to model geographic features; for example, arcs and nodes can be used to model linear features such as street centerlines. The tic, annotation, link, and boundary feature classes provide supporting data for coverage data management and viewing.
2. The conceptual representation of a geographic feature. When referring to geographic features, feature classes include point, line, area, and surface.

Feature Dataset
A feature dataset is a collection of feature classes that share the same spatial reference.

Field Links
Define the actual mapping between model element attributes and columns within each database table.

File Extension
The period and three characters, typically, at the end of a filename. A file extension usually identifies the kind of information the file contains. For example, files you create in AutoCAD have the extension *.DWG.

Fire Flow Upper Limit
The maximum allowable fire flow that can occur at a withdrawal location. This is a user-specified practical limit that will prevent the program from computing unrealistically high fire flows at locations such as primary system mains, which have large diameters and high service pressures. Remember that a system's ability to deliver fire flows is ultimately limited by the size of the hydrant opening and service line, as well as the number of hydrants available to combat a fire at a specific location.

Flow
Represents the calculated value of the pipe, valve, or pump discharge at the given time.

From Node
Represents the pipe's starting node. Positive flow rates are in the direction of the “From”node towards the “To Node”. Negative flow rates are in the opposite direction.

From Pipe
The pipe that connects to the upstream side of a valve or pump.

G

GA
Genetic algorithm.

GEMS Datastore
The relational database that Bentley WaterGEMS V8*i* uses to store model data. Each Bentley WaterGEMS V8*i* project uses two main files for data storage, the datastore (.sqlite) and the Bentley WaterGEMS V8*i* Modeler-specific data (.wtg). Although the Bentley WaterGEMS V8*i* datastore is an .sqlite file, **cannot** be a geodatabase.

Generations
The maximum value for genetic algorithm generations is determined by the Maximum Era Number and Era Generation Number you set in the GA Parameters. The actual number of generations that get calculated depend on the Stopping Criteria you set.

Geocode
The process of identifying the coordinates of a location given its address. For example, an address can be matched against a TIGER street network to determine the location of a home. Also referred to as address geocoding.

Geodatabase
Short for geographic database, a geodatabase stores spatial and descriptive data in an efficient manner. Geodatabases are the standard file format for ArcGIS v8 and later.

H

Headloss
Represents the energy lost due to friction and minor losses. The headloss field displays the pipe, valve, or pump's total headloss at the given time.

Headloss Gradient
Presents the headloss in the pipe as a slope, or gradient. This allows you to more accurately compare headlosses for pipes of different lengths.

Hydraulic Grade
Elevation to which water would rise under zero pressure. For open surfaces, such as reservoirs and tanks, this is equal to the water surface elevation. The hydraulic grade field presents the hydraulic grade for the element at the current time period as calculated based on the system flow rates and head changes.

Hydraulic Grade Setting
The constraint to which a valve regulates, expressed in units of head (Length). Depending on the type of valve, it may refer to either the upstream or downstream hydraulic grade or the headloss across the valve.

I

Inactive Volume
The volume of water below the minimum elevation of the tank. This volume of water is always present, even when the tank reaches its minimum elevation and closes itself off from the system. Therefore, it is hydraulically inactive. It is primarily used for water quality calculations.

Inflow & Outflow
An inflow is a flow into a node from the system, while an outflow is a flow from the node into the system. A negative outflow is the same as a positive inflow, and a negative inflow is the same as a positive outflow.

Inheritance
Refers to the parent-child relationships used by scenarios and alternatives. Just as in the natural world, inheritance is used to refer to the situation where an entity receives something from its parent. For example, we speak of a child inheriting blue eyes from a parent. Unlike in the natural world, inheritance in scenarios and alternatives is dynamic. If the parent's attribute changes, the child's attribute automatically changes at the same time, unless the value is explicitly changed in a child.

Initial Settings
Sets the status of an element for a steady-state analysis or the first time step in an extended period simulation. The initial settings for a pipe, pump, or valve can be set using the elemental dialog boxes or a table.

Initial Water Quality
Represents the starting conditions at a node for age, trace, or constituent concentration. The initial value will be slightly different depending on the analysis type.

Interpolate
Estimating a value between two known values assuming a linear relationship. See also: extrapolate.

Invert
Lowest point of a pipe opening. Sometimes referred to as the flow line.

L

Label
The unique name by which an element will be referenced in reports, error messages, and tables.

Layer
Layers contain spatial data according to similar subject matter. Conceptually, layers in a database or map library environment are exactly like coverages. Layers are the standard GIS data format for ArcView 3.x and earlier.

Length
Represents the distance on a pipe from the From Node to the To Node, according to the scaled length of the pipe. To enter an overriding length, click the **User Defined Length** field and type in your desired length value.

LIDAR
Light Detection and Ranging.

M

Manning's Coefficient
Roughness coefficient used in Manning's Formula.

Material
The selection of a pipe's construction material. This material will be used to determine a default value for the pipe's roughness.

Maximum Elevation
The highest allowable water surface elevation in a tank. If the tank fills above this point, it will automatically shut off from the system.

Max. Extended Operating Point
The absolute maximum discharge at which a pump can operate, with zero head being added to the system. This value may be computed by the program or entered manually.

Maximum Operating Point
The highest discharge for which a pump is actually intended to run. At discharges above this point, the pump may behave unpredictably or its performance may decline rapidly.

Menu
A menu of available commands or actions you can perform. Access menus from the menu bar at the top of the main program window.

Messages
The section that contains information generated during the calculation of the model, such as warnings, errors, and status updates.

Messages Light
A light that appears on the Tab of the Messages sheet. The light will be red if errors occurred during the analysis, yellow if there are warnings or cautions, and green if there are no warnings or errors.

Metadata
Additional information (aside from tabular and spatial data) that makes the data useful. Includes characteristics and information that are required to use the data but are not contained within the data itself.

Minimum Elevation
The lowest allowable water surface elevation in a tank. If the tank drains below this point, it will automatically shut off from the system.

Minimum System Junction
The junction where the calculated minimum system pressure occurs.

Minimum System Pressure
The minimum pressure allowed at any junction in the entire system as result of fire flow withdrawal. If the pressure at a node anywhere in the system falls below this constraint while withdrawing fire flow, fire flow will not be satisfied. A fire flow analysis may be configured to ignore this constraint.

Minimum Zone Junction
The junction where the calculated minimum zone pressure occurs.

Minimum Zone Pressure
The minimum pressure to maintain at all junction nodes within a Zone. The model determines the available fire flow such that the minimum zone pressures do not fall below this target pressure. Each junction has a zone associated with it, which can be specified in the junction's input data. If you do not want a junction node to be analyzed as part of another junction node's fire flow analysis, move it to another Zone.

Minor Loss
The field that presents the total minor loss K value for a pipe or valve. If an element has more than one minor loss, each can be entered individually by clicking the **Ellipsis (...)** button.

Modeler/Stand-Alone
The Bentley software environment, and not the AutoCAD or ESRI one.

Mouse Buttons
The left mouse button is the primary button for selecting or activating commands. The right mouse button is used to activate shortcut context menus and help. Note that the mouse button functions can be redefined using the Windows Control Panel. If your mouse is equipped with a mouse wheel, you can use it for various panning and zooming functions.

N

.nrg
File containing energy cost results.

Needed Fire Flow
The flow rate required at a junction to satisfy fire flow demands.

Network Element
An element that forms part of the network model. Annotation elements, such as polylines, borders, and text, are not network elements.

Number
The number of parallel conveyance elements in a model.

Notes
The field that allows you to enter text relevant to the model. It may include a description of an element, a summary of your data sources, or any other information of interest.

O

.out
File with complete scenario results.

ODBC
Open Database Connectivity (ODBC) is a standard programming interface developed by Microsoft for accessing data in relational and non-relational database management systems (DBMS).

On/Off Status
The status of a pump can be either on or off. On means that flow will occur in the downstream direction, and the pump will add head to the system according to its characteristic curve. Off means that no flow will occur, and no head will be added.

Open/Closed Status
The status of a pipe can be either open or closed. Open means that flow can occur in either direction. Closed means that no flow will occur through the pipe.

P

.pv8
The previous version for files upgraded to new.

PBV
Pressure breaker valve.

Percent Full
The ratio of the current storage volume to the total storage volume, multiplied by 100.

Pipe Status
Indicates whether the pipe is open or closed. As input, this determines how the pipe begins the simulation. As output, it shows the calculated status of the pipe at the given time.

Polyline
A composite element that consists of a series of line segments. Each line segment begins and ends at a vertex. A vertex may be another element such as a junction, tank, or pump.

Power
Represents the water horsepower of a pump. This is the horsepower that is actually transferred from the pump into the water. Depending on the pump's efficiency, the actual power consumed (brake horsepower) may vary.

Pressure
The field that displays the pressure for the current time period.

Pressure Setting
The constraint to which a valve regulates, expressed in units of pressure (Force per Length^2). Depending on the type of valve, it may refer to either the upstream or downstream pressure or the pressure drop.

PRV
Pressure reducing valve.

PSV
Pressure sustaining valves.

Pump Status
A pump can have two different status conditions: On, which is normal operation; or Off, which is no flow under any condition.

R

.rpc
The file with scenario messages.

RDBMS
An acronym that stands for Relational Database Management System.

Relate
A temporary connection between table records using a common item shared by both tables. Each record in one table is connected to those records in the other table that share the same value for the common item.

Relational Database
A database in which the data is structured in such a way as to associate tables according to attributes that are shared by the tables.

Relational Join
The process of merging two attribute tables using a common item.

Relative Speed Factor
Defines the characteristics of a pump relative to the speed for which the pump curve was entered, in accordance with the affinity laws. A speed factor of 1.00 would indicate pump characteristics identical to those of the original pump curve.

Residual Pressure
The minimum residual pressure to occur at a junction node. The program determines the amount of fire flow available such that the residual pressure at a junction node does not fall below this target pressure.

Reynolds Number
Ratio of viscous forces relative to inertial forces. A high Reynold's number indicates turbulent flow, while a low number indicates laminar flow.

Roughness
A measure of a pipe's resistance to flow. Pipes of different ages, construction material, and workmanship may have different roughness values.

Roughness Coefficient
A value used to represent the resistance of a conveyance element to flow. In the Manning's equation, this value is inversely proportional to flow. The smaller the roughness coefficient, the greater the flow.

S

.**sqlite**
The open database file.

Satisfies Fire Flow
A true or false statement indicating whether this junction node meets the fire flow constraints. A check mark in the box means the Fire Flow Constraints were satisfied for that node. If there is no check mark, the Fire Flow Constraints were NOT satisfied.

Schema
A diagrammatic representation; an outline or model. Essentially, a schema represents the number of tables, the columns they contain, the data types of the columns, and any relationships between the tables.

Select
The process of adding one or more elements to an active selection set.

Selection Set
The active group of selected elements. A selection set allows editing or an action, such as move or delete, to be performed on a group of elements.

Shape
The cross-sectional geometric form of a conveyance element (i.e., circular, box, arch, etc.).

Shapefile
A file format that stores spatial and attribute data for the spatial features within the dataset. A shapefile consists of a main file, an index file, and a dBASE table. Shapefiles were the standard file storage format for ArcView 3.x and earlier.

Shutoff Point
The point at which a pump will have zero discharge. Typically the maximum head point on a pump curve.

Size
Inside diameter of a pipe section for a circular pipe.

Spatial Reference
The spatial reference for a feature class describes its coordinate system (for example, geographic, UTM, and State Plane), its spatial domain, and its precision. The spatial domain is best described as the allowable coordinate range for x, y coordinates, m- (measure) values, and z-values. The precision describes the number of system units per one unit of measure. A spatial reference with a precision of 1 will store integer values, while a precision of 1000 will store three decimal places.

Stand-Alone/Modeler
The Bentley Systems software environment, and not the AutoCAD or ESRI one.

Starting Elevation
The value that is used as the beginning condition for an extended period simulation.

Status Pane
The area at the bottom of the window used for displaying status information.

Storage Node
Special type of node where a free water surface exists, and the hydraulic head is the elevation of the water surface above sea level.

T

Table Links
A table link must be created for every database table or spreadsheet worksheet that is to be linked to the current model. Any number of Table Links may reference the same database file.

TCV
Throttle control valve.

To Node
Represents a pipe's ending node. Positive flow rates are in the direction of from towards to. Negative flow rates are in the opposite direction.

To Pipe
The pipe that connects to the downstream side of a valve or pump.

Total Active Volume
The volume of water between minimum elevation and maximum elevation of a tank. This is an input value for variable area tanks.

Total Storage Volume
The holding capacity of a tank. It is the sum of the maximum hydraulically active storage volume and the hydraulically inactive storage volume.

Total Needed Fire Flow
If you choose to add the fire flow to the baseline demand, the Total Needed Fire Flow is equal to the Needed Fire Flow plus the baseline demand. If you choose not to add the fire flow to the baseline demand, the Total Needed Fire Flow is equal to the Needed Fire Flow.

Trace (Source Ident.)
Determines what percentage of water at any given point originated at a chosen tank, reservoir, or junction.

Trials
The maximum value for genetic algorithm trials is determined by what you set for Stopping Criteria. Note that you can set a number larger than (Maximum Era Number)*(Era Generation Number)*(Population Size), but calculations beyond that number (for this example, the value is 45,000) are less likely to produce significant improvements in optimization.

V

Valve Status
A valve can have several different status conditions: Closed (no flow under any condition), Active (throttling, opening, or closing dependent on system pressures and flows), and Inactive (wide open, with no regulation).

Velocity
The field that displays the calculated value for a pipe, valve, or pump velocity at a given time. It is found by dividing the element's flow rate by its cross-sectional area.

Vertex
An element in a topological network.

W

.wtg
File that displays WaterGEMS V8*i* information.

wtg.sqlite
To distinguish between the WaterGEMS V8*i* modeling data file and another programs data file. The most important file because it contains all of the modeling data.

Wall Reaction Coefficient
Defines the rate at which a substance reacts with the wall of a pipe, and is expressed in units of length/time.

Bentley WaterGEMS V8*i* Datastore
The relational database that Bentley WaterGEMS V8*i* uses to store model data. Each Bentley WaterGEMS V8*i* project uses two main files for data storage, the datastore (.sqlite) and the Bentley WaterGEMS V8*i* specific data (.wtg).

WaterGEMS V8i File Types

The following lists different types of files that can be used with WaterGEMS V8i.
.bak – backup of most files
GEMS Data Store – modeling data
Geodatabase – topology (in ArcGIS version)
.dwh, .dgn, .dwg – drawing information in stand-alone, Microstation, AutoCAD
.out – complete results by scenario
.rpc – scenario messages
.nrg – energy cost results
.pv8 – previous version for files upgraded to new
.xml – used for libraries

WaterObjects
The object model used by Bentley WaterGEMS V8*i,* which allows for the extension and customization of the core software functions.

Water Quality
The field that displays the water quality for the current time period.

Water Quality Analysis
An analysis that can be one of three types: Age, Trace, or Constituent.

X

.**xml**
File used for libraries.

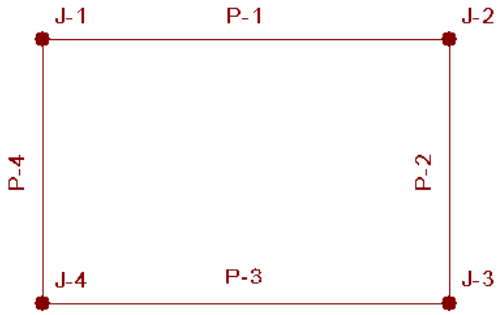
1 INTRODUCTION

1.1 What is a Water Distribution System Computer Model?

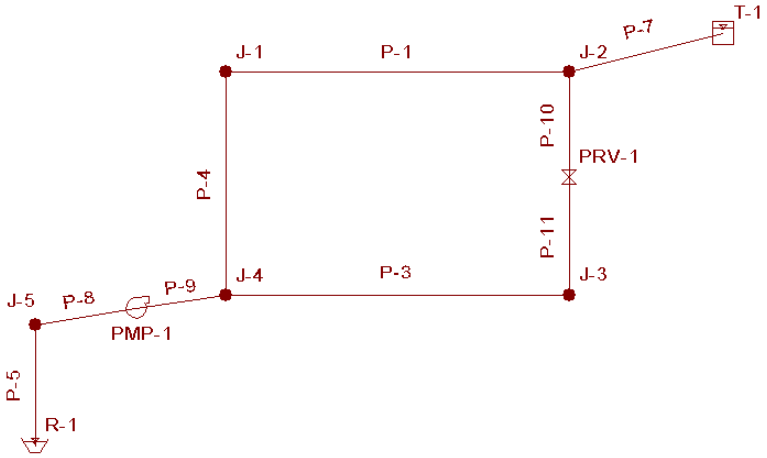
- A mathematical representation of a real physical water distribution system.
- Predicts flow of water through pipes and pressures at junctions of pipes.

1.2 What are the Components in a Computer Model?

- Pipes with length, diameter, roughness, open/close information.
- Junctions with elevation, inflow and/or demand data.

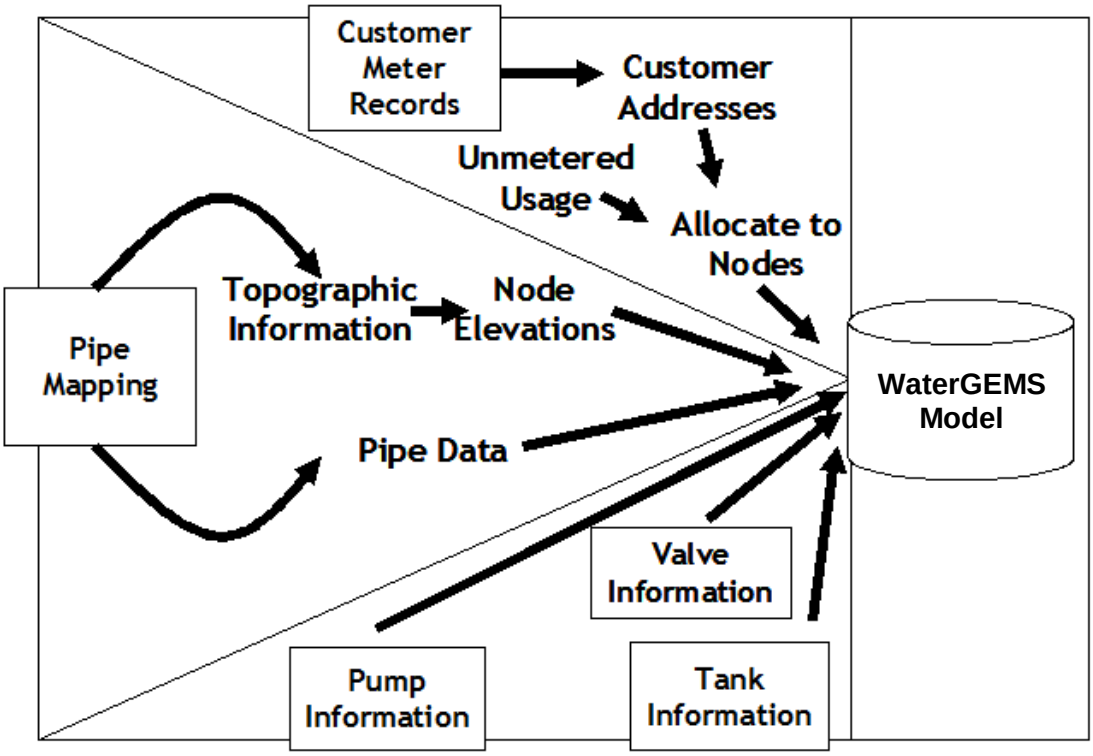


- Reservoirs.
- Tanks with elevation, dimensional data.
- Pumps with characteristics like head-discharge relationship, on/off status.
- Valves performing special functions like pressure regulation, flow control, etc.



1.3 What Can Computer Models Do?

- Identify system deficiencies like high/low pressures, high head loss, and inadequate storage.
- Analyze the impact of proposed development or long-term growth within the system.
- Determine the ability of the system to deliver adequate fire flows.
- Size pipes, select pumps, locate tanks.
- Evaluate operating strategies.
- Assess pipeline rehabilitation methods.
- Develop Capital improvement plans.
- Develop emergency response plans.
- Estimate the quality of water throughout the system.



1.4 Basic Steps for Model Development

- Collect data on Pipes: Maps, Age, Material.
- Collect Data on Junctions: Elevations.
- Categorize and Allocate Demands: Metered, Unmetered.
- Define System's Pumps, Valves and Tanks.
- Define water supply sources.

1.5 Steps to Complete Model

- Fix and confirm network connectivity.
- Correct calculation errors.
- Define modeling scenarios.
- Calibrate the model.
- Verify the calibrated model.

1.6 What Type of Simulations Can a Model Perform?

- Various Demand Conditions such as Average Day, Maximum Day, Peak Hour.
- Water quality analysis of water age, source tracing, and/or constituent growth/decay.
- Energy use analysis.
- System curve analysis.
- Hydrant curve definition.

- Steady-State Hydraulic analysis for a "snapshot" view of the system.
- Fire Flow Analysis to Assess Extreme Conditions.
- Extended-Period Simulation to see how the system behaves over time.

2 MODELINGBASICS

2.1 Basic Working Equations

- Conservation of Mass (Flow Continuity)

  $Q_{in} = Q_{out} + Q_{demand}$
- Conservation of Energy (Head Loss Continuity)

  $Z_1 + P_1/\gamma + V_1^2/2g = Z_2 + P_2/\gamma + V_2^2/2g + \sum h_L - \sum hp$

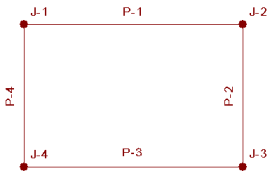
  $\sum h_L$: the sum of friction and minor losses.
  Σh_p: Head gain from a pump

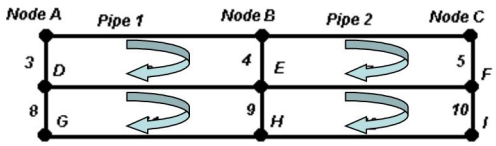
2.2 Units of Pressure and Flow

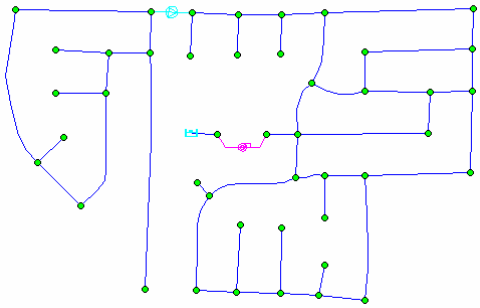
- Pressure: 1 kPa = 0.10197 m H_2O
- Flow: 1 ML/day = 0.001 m^3/day = 11.574 L/s = 0.01157 m^3/s

2.3 Solution Methods

- Balance flows and pressures in pipe loops using basic equations.
- Results from pipes and nodes common to more than one loop must be equal.
- System divided into matrix forms which are groups of pipe loops.
- Solve the system of equations in matrix form numerically following an iterative solution scheme until the differences between solutions falls within a specified tolerance.



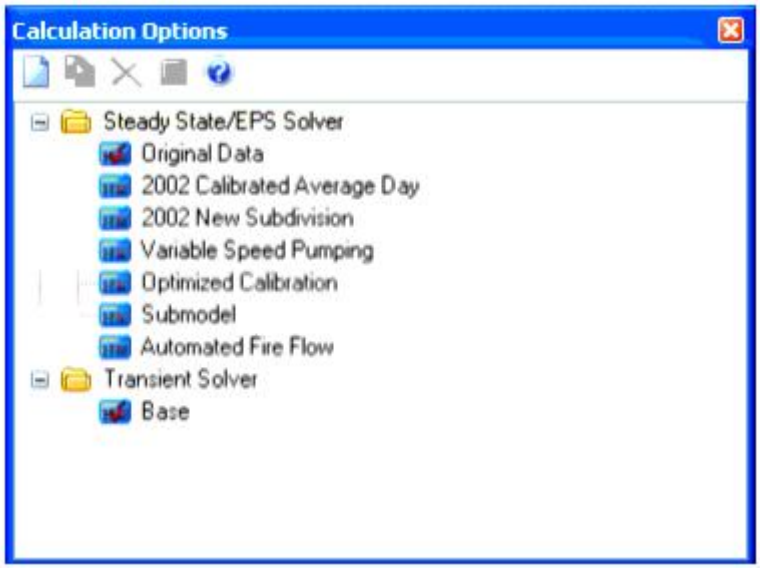




2.4 Calculation Options

Calculations depend on a variety of parameters that may be configured by the user.

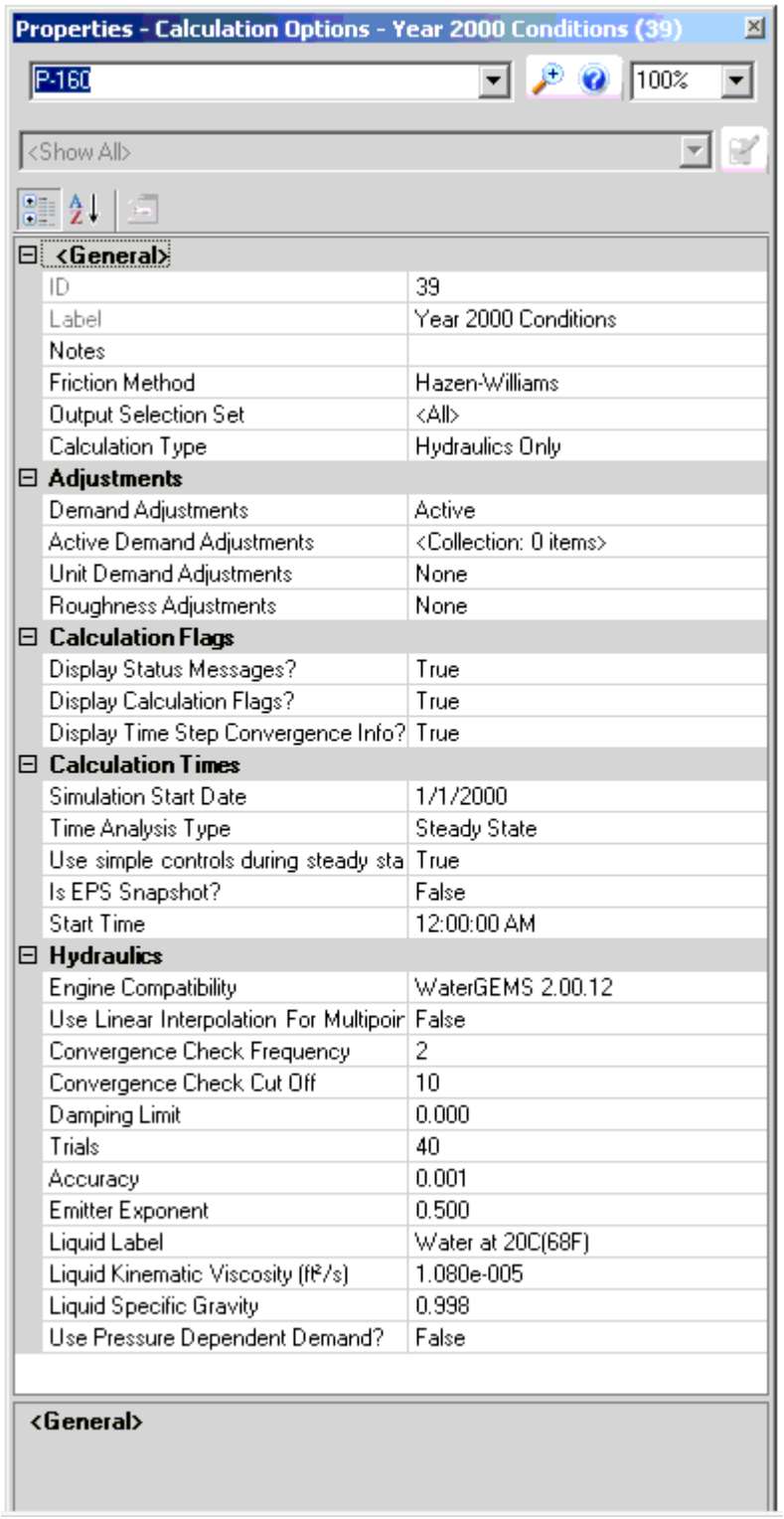
Choose Analysis > Calculation Options, Alt+3, or click the button to open the Calculations Options dialog box.



New	Creates a new calculation option.
Duplicate	Makes a copy of the selected calculation option.
Delete	Deletes the selected calculation option. The base calculation option cannot be deleted.
Rename	Renames the selected calculation option.
Help	Displays online help for the Calculation Options.

The following controls are available from the Calculation Options dialog box.

To view the Steady State/EPS Solver properties of the Base Calculation Options:
Select Base Calculation Options under Steady State/EPS Solver and double click to open the Properties dialog box.



The following calculation option parameters are available for user configuration:

- **Friction Method** – Set the global friction method.
- **Output Selection Set** – Select whether to generate output for All Elements (the default setting) or only the elements contained within the chosen selection set.
- **Calculation Type** – Select the type of analysis to perform with this calculation options set.

- **Consider Pumps and Valves in Min. System Pressure Constraints?** –If True, the pressures at pumps and valves will be considered.
- **Demand Adjustments** –Specify whether or not to apply adjustment factors to standard demands.
- **Active Demand Adjustments –** The collection of demand adjustments that are applied during the analysis.
- **Unit Demand Adjustments –** Specify whether or not to apply adjustment factors to unit demands.
- **Active Unit Demand Adjustments** – The collection of unit demand adjustments that are applied during the analysis.
- **Roughness Adjustment** – Specify whether or not to apply adjustment factors to roughnesses.
- **Active Roughness Adjustments** – The collection of roughness adjustments that are applied during the analysis.
- **Display Status Messages**? – If set to True, element status messages will be stored in the output and reported.
- **Display Calculation Flags?** – If set to True, calculation flags will be stored in the output and reported.
- **Display Time Step Convergence Info?** – If set to True, convergence/iteration data for each time step will be stored in the output file and displayed in the calculation summary.
- **Simulation Start Date** – Select the calendar date on which the simulation begins.
- **Time Analysis Type** – Select whether the analysis is extended period or steady-state.
- **Use simple controls during steady state?** – When True, simple controls will be active during steady state analyses, else they will not be used. Note that logical controls are never used during steady state analysis.
- **Is EPS Snapshot?** – If True then an EPS snapshot is run instead of a regular steady state run. An EPS snapshot is a steady state run, but it considers the starting date and time of analysis and applies the appropriate pattern multipliers for that time. Note that since an EPS is not run, attributes such as tank levels are derived from the same initial conditions as a steady state run.
- **Equivalent Hydraulic Time Step** – In order that the pattern multipliers used in an EPS snapshot run exactly match those in an equivalent EPS run, specify the hydraulic time step of the EPS run that you wish to match.
- **Start Time** – Select the clock time at which the simulation begins.
- **Duration** – Specify the total duration of an extended period simulation.
- **Hydraulic Time Step** – Select the length of the calculation time step.
- **Override Reporting Time Step?** – Specify if you want the Reporting Time Step to differ from the Hydraulic Time Step.
- **Reporting Time Step** – Data will be presented at every reporting time step. The reporting time step should be a multiple of the hydraulic time step.
- **Set Water Quality Time Step?** – If set to True, the Water Quality Time Step can be adjusted, otherwise it is computed by the calculation engine. It is not recommended that you set this to True.
- **Water Quality Time Step** – Time interval used to track water quality changes throughout the network. By default, this value is computed by the numerical engine and is equivalent to the smallest travel time through any pipe in the system.
- **Engine Compatibility** – This field allows you to choose which engine compatibility mode you want to run in. Choose WaterGEMS 2.00.12 to get all of the latest engine improvements and fixes made by Bentley and an engine mode that is based upon EPANET 2.00.12. This is the default setting for new models. Choose WaterGEMS 2.00.10 to maintain compatibility with previous

version of WaterGEMS (V8i SELECTseries 1 and earlier), where the computational engine is based on EPANET 2.00.10. This is the default for upgraded models. If you select one of the EPANET modes, any enhancements, calculation corrections, and bug fixes made by Bentley will be disabled in order to match EPANET version results. Imported EPANET models will default to the appropriate EPANET version.

- **Use Linear Interpolation for Multipoint Pumps?** – If set to true the engine will use linear interpolation to interpret the pump curve as opposed to quadratic interpolation.
- **Convergence Check Frequency** – This option sets the number of solution trials that pass during hydraulic balancing before the status of pumps, check valves, flow control valves, and pipes connected to tanks is updated. The default value is 2, meaning that status checks are made every other trial. A value equal to the maximum number of trials would mean that status checks are made only after the system has converged. (Whenever a status change occurs the trials must continue since the current solution may not be balanced.) The frequency of status checks on pressure reducing and pressure sustaining valves is determined by the Damping Factor calculation option.
- **Convergence Check Cut Off** – This option is the number of solution trials after which periodic status checks on pumps, check valves, flow control valves, and pipes connected to tanks are discontinued. Instead, a status check is made only after convergence is achieved. The default value is 10, meaning that after 10 trials, instead of checking status at every trial indicated by the Convergence Check Frequency setting, status is checked only at convergence.
- **Damping Limit** – This is the accuracy value at which solution damping and status checks on PRVs and PSVs should begin. Damping limits all flow changes to 60 percent of what they would otherwise be as future trials unfold. The default of 0 indicates that no damping should be used and that status checks on control valves are made at every iteration. Damping might be needed on networks that have trouble converging, in which case a limit of 0.01 is suggested (relative to the default calculation hydraulic accuracy of 0.001).
- **Trials** – Unit-less number that defines the maximum number of iterations to be performed for each hydraulic solution. The default value is 40.
- **Accuracy** – Unit-less number that defines the convergence criteria for the iterative solution of the network hydraulic equations. When the sum of the absolute flow changes between successive iterations in all links is divided by the sum of the absolute flows in all links and is less than the Accuracy, the solution is said to have converged. The default value is 0.001 and the minimum allowed value for Accuracy is 1.0e-5.
- **Emitter Exponent** – Emitters are devices associated with junctions that model the flow through a nozzle or orifice. In these situations, the demand (i.e., the flow rate through the emitter) varies in proportion to the pressure at the junction raised to some power. The constant of proportionality is termed the discharge coefficient. For nozzles and sprinkler heads the exponent on pressure is 0.5 and the manufacturer usually states the value of the discharge coefficient as the flow rate in gpm through the device at a 1 psi pressure drop.
- **Liquid Label** – Label that describes the type of liquid used in the simulation.
- **Liquid Kinematic Viscosity** – Ratio of the liquid's dynamic or absolute viscosity to its mass density.
- **Liquid Specific Gravity** – Ratio of the specific weight of the liquid to the specific weight of water at 4 degrees C or 39 degrees F.
- **Use Pressure Dependent Demand?** – If set to True the flows at junctions and hydrants will be based on pressure constraints.

- **Age Tolerance** – If the difference between two parcels of water is equal to or less than the value specified in this field, the parcels are considered to be of equal age.
- **Constituent Tolerance** – If the difference between two parcels of water is equal to or less than the value specified in this field, the parcels are considered to possess an equal concentration of the associated constituent.
- **Trace Tolerance** – If the difference between two parcels of water is equal to or less than the value specified in this field, the parcels are considered to be within the same percentile.

2.5 Trials

Unit-less number that defines the maximum number of iterations to be performed for each hydraulic solution. The default value is 40.

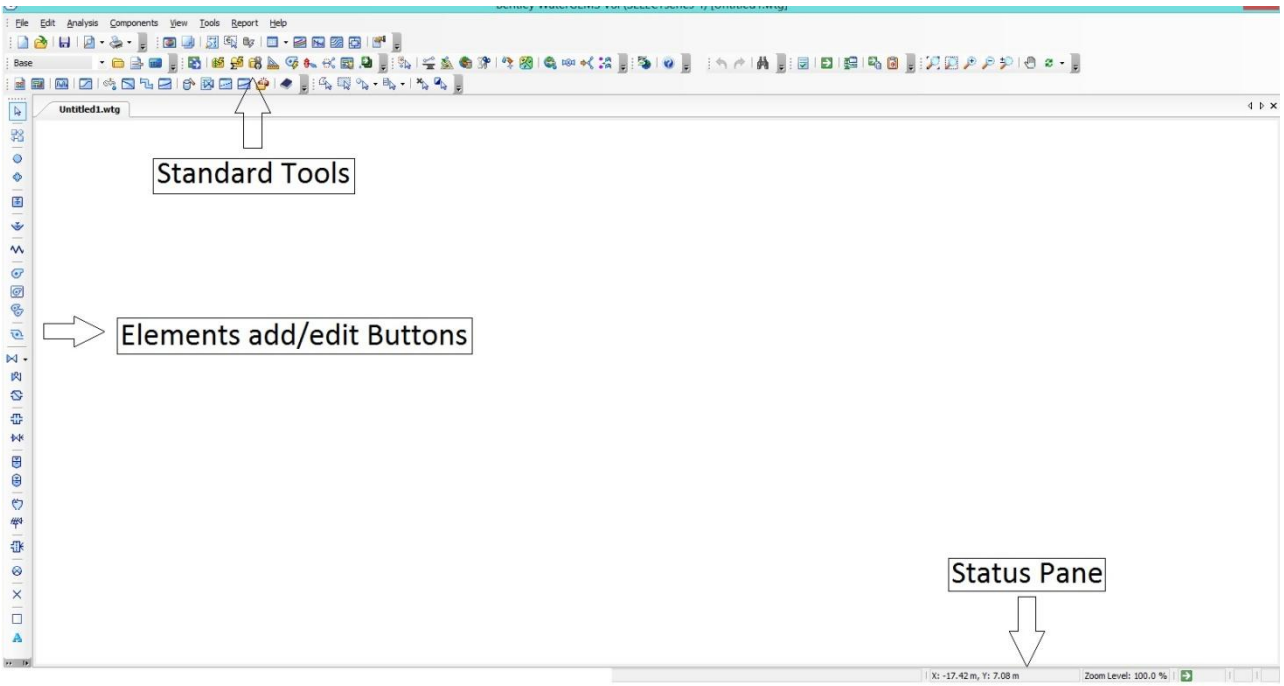
2.6 Accuracy

Unit-less number that defines the convergence criteria for the iterative solution of the network hydraulic equations. When the sum of the absolute flow changes between successive iterations in all links is divided by the sum of the absolute flows in all links, and is less than the Accuracy, the solution is said to have converged. The default value is 0.001 and the minimum allowed value for Accuracy is 1.0e-5.

3 WATERGEMSV8I FEATURES

WaterGEMS runs on your choice of platforms: Standalone, integrated with GIS, integrated with AutoCAD, integrated with Microstation.

3.1 Basic WaterGEMS interface



3.2 Zooming

You can enlarge or reduce your model in the drawing pane using one of the following zoom tools:

The current zoom level is displayed in the lower right hand corner of the interface, next to the coordinate display.

Zoom Extents 

The Zoom Extents command automatically sets the zoom level such that the entire model is displayed in the drawing pane.

To use Zoom Extents, click Zoom Extents on the Zoom toolbar. The entire model is displayed in the drawing pane.

or

Select **View > Zoom > Zoom Extents**.

Zoom Window

The Zoom Window command is used to zoom in on an area of your model defined by a window that you draw in the drawing pane.

To use Zoom Window, click the Zoom Window button on the Zoom toolbar, then click and drag the mouse inside the drawing pane to draw a rectangle. The area of your model inside the rectangle will appear enlarged.

or

Select **View > Zoom > Zoom Window**, then draw the zoom window in the drawing pane.

Zoom In and Out

The Zoom In and Zoom Out commands allow you to increase or decrease, respectively, the zoom level of the current view by one step per mouse click.

To use Zoom In or Zoom Out, click either one on the Zoom toolbar, or select **View > Zoom > Zoom In** or **View > Zoom > Zoom In**.

If your mouse is equipped with a mousewheel, you zoom in or out by simply moving the mousewheel up or down respectively.

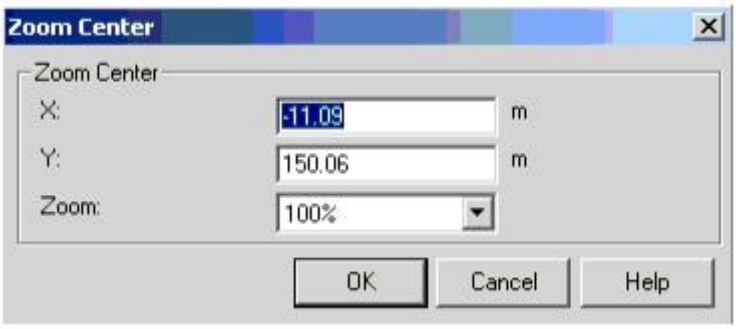
Zoom Realtime

The Zoom Realtime command is used to dynamically scale up and down the zoom level. The zoom level is defined by the magnitude of mouse movement while the tool is active.

Zoom Center

The Zoom Center command is used to enter drawing coordinates that will be centered in the drawing pane.

1. Choose View > Zoom > Zoom Center or click the Zoom Center icon on the Zoom toolbar.. The Zoom Center dialog box opens.



2. The Zoom Center dialog box contains the following:

X	Defines the X coordinate of the point at which the drawing view will be centered.
Y	Defines the Y coordinate of the point at which the drawing view will be centered.
Zoom	Defines the zoom level that will be applied when the zoom center command is initiated. Available zoom levels are listed in percentages of 25, 50, 75, 100, 125, 150, 200 and 400.

3. Enter the X and Y coordinates.
4. Select the percentage of zoom from the Zoom drop-down menu.
5. Click OK.

Zoom to Selection

Enables you to zoom to specific elements in the drawing. You must select the elements to zoom to before you select the tool.

Zoom Previous and Zoom Next

Zoom Previous returns the zoom level to the most recent previous setting. To use Zoom Previous, click View > Zoom > Zoom Previous or click the Zoom Previous icon from the Zoom toolbar.

Zoom Next returns the zoom level to the setting that was active before a Zoom Previous command was executed. To use Zoom Previous, click View > Zoom > Zoom Next or click the Zoom Next icon from the Zoom toolbar.

3.3 Panning

You can change the position of your model in the drawing pane by using the Pan tool.

To use the Pan tool

1. Click the **Pan**  button on the Zoom toolbar.

   The mouse cursor changes to the Pan icon.

2. Click anywhere in the drawing, hold down the mouse button and move the mouse to reposition the current view.

   or

   If your mouse is equipped with a mousewheel, you can pan by simply holding down the mousewheel and moving the mouse to reposition the current view.

   or

   Select **View > Pan**, then click anywhere in the drawing, hold down the mouse button and move the mouse to reposition the current view.

3.4 MicroStation Environment

In the MicroStation environment you can create and model your network directly within your primary drafting environment. This gives you access to all of MicroStation's powerful drafting and presentation tools, while still enabling you to perform Bentley WaterGEMS V8*i* modeling tasks like editing, solving, and data management. This relationship between Bentley WaterGEMS V8*i* and MicroStation enables extremely detailed and accurate mapping of model features, and provides the full array of output and presentation features available in MicroStation. This facility provides the most flexibility and the highest degree of compatibility with other CAD-based applications and drawing data maintained at your organization.

Bentley WaterGEMS V8*i* features support for MicroStation integration. You run Bentley WaterGEMS V8*i* in both MicroStation and stand-alone environment.

The MicroStation functionality has been implemented in a way that is the same as the Bentley WaterGEMS V8*i* base product. Once you become familiar with the stand-alone environment, you will not have any difficulty using the product in the MicroStation environment.

In the MicroStation environment, you will have access to the full range of functionality available in the MicroStation design and drafting environment. The standard environment is extended and enhanced by using MicroStation's MDL (MicroStation Development Language) client layer that lets you create, view, and edit the native Bentley WaterGEMS V8*i* network model while in MicroStation.

MDL is a complete development environment that lets applications take full advantage of the power of MicroStation and MicroStation-based vertical applications. MDL can be used to develop simple utilities, customized commands or sophisticated commercial applications for vertical markets.

Some of the advantages of working in the MicroStation environment include:

- Lay out network links and structures in fully-scaled environment in the same design and drafting environment that you use to develop your engineering plans.

- Have access to any other third party applications that you currently use, along with any custom MDL applications.
- Use native MicroStation insertion snaps to precisely position Bentley WaterGEMS V8*i* elements with respect to other entities in the MicroStation drawing.
- Use native MicroStation commands on Bentley WaterGEMS V8*i* model entities with automatic update and synchronization with the model database.
- Control destination levels for model elements and associated label text and annotation, giving you control over styles, line types, and visibility of model elements.

*Note: Bentley MicroStation V8i is the only MicroStation environment supported by WaterGEMS V8*i*.*

Additional features of the MicroStation version includes:

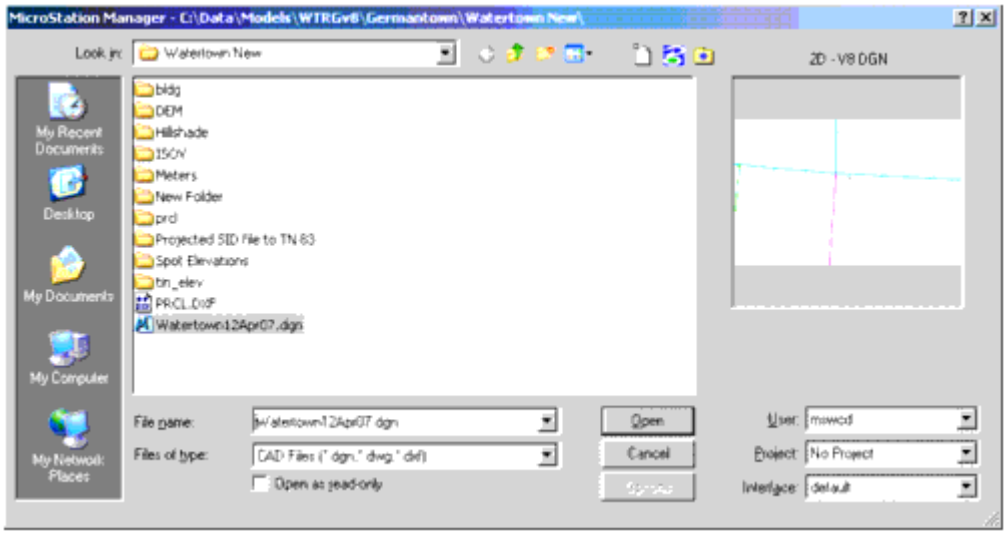
- MicroStation Project Files
- Bentley WaterGEMS V8i Element Properties
- Working with Elements
- MicroStation Commands
- Import Bentley WaterGEMS V8i

3.5 Getting started in the MicroStation environment

A Bentley MicroStation WaterGEMS V8*i* project consists of:

- **Drawing File (.DGN)** – The MicroStation drawing file contains the elements that define the model, in addition to the planimetric base drawing information that serves as the model background.
- **Model File(.**wtg**)** – The model file contains model data specific to WaterGEMS V8*i*, including project option settings, color-coding and annotation settings, etc. Note that the MicroStation .dgn that is associated with a particular model may not necessarily have the same filename as the model's .wtg file.
- **Database File (.sqlite)** – The model database file that contains all of the input and output data for the model. Note that the MicroStation .dgn that is associated with a particular model may not have the same filename as the model's .sqlite file.

When you start Bentley WaterGEMS V8*i* for MicroStation, you will see the dialog below. You must identify a new or existing MicroStation dgn drawing file to be associated with the model before you can open a Bentley WaterGEMS V8*i* model.



Either browse to an existing dgn file or create a new file using the new button on the top toolbar. Once you have selected a file, you can pick the Open button.

Once a drawing is open, you can use the WaterGEMS V8*i* Project drop down menu to create a new WaterGEMS V8*i* project, attach an existing project, or import a project.

There are a number of options for creating a model in the MicroStation client:

- **Create a model from scratch** – You can create a model in MicroStation. You'll first need to create a new MicroStation .dgn (refer to your MicroStation documentation to learn how to create a new .dgn). Start WaterGEMS V8*i* for MicroStation. In the first dialog, pick the New button and assign a name and path to the DGN file. Once the dgn is open, use the New command in the WaterGEMS V8*i* Project menu (Project > New). This will create a new WaterGEMS V8*i* project file and attach it to the Bentley MicroStation .dgn file. Once the file is created you can start creating WaterGEMS V8*i* elements that exist in both the WaterGEMS V8*i* database and in the .dgn drawing. See Working with Elements and Working with Elements Using MicroStation Commands for more details.
- **Open a previously created** WaterGEMS V8*i* **project** – You can open a previously created WaterGEMS V8*i* model and attach it to a .dgn file. To do this, start WaterGEMS V8*i* for MicroStation. Open or create a new MicroStation .dgn file (refer to your MicroStation documentation to learn how to create a new .dgn). Use the Project menu on the WaterGEMS V8*i* toolbar and click on the Project > "Attach Existing..." command, then select an existing WaterGEMS V8*i*.wtg file. The model will now be attached to the .dgn file and you can edit, delete, and modify the WaterGEMS V8*i* elements in the model. All MicroStation commands can be used on WaterGEMS V8*i* elements.
- **Import a model that was created in another modeling application** – There are four types of files that can be imported into WaterGEMS V8*i*:
    - **WaterGEMS** / **WaterCAD** / **HAMMER Database** – this can either be a HAMMER V8*i* or V8, WaterGEMS V8*i* or V3, or WaterCAD V8*i* or V7 database. The model will be processed and imported into the active MicroStation .dgn drawing. See Exporting a HAMMER v7 Model for more details.
    - **EPANET** – You can import EPANET input (.inp) files. The file will be processed and the proper elements will be created and added to the

MicroStation drawing. See Importing and Exporting EPANET Files for more details.

- **Submodel** – You can import a WaterGEMS V8*i* V8 subenvironment into the MicroStation drawing file. See Importing and Exporting Submodel Files for more details.

If you want to trace the model on top of a dgn or other background file, you would load the background into the dgn first by using either File/Reference or File/Raster Manager Then you start laying out elements over top of the background.

3.6 The MicroStation Environment Graphical Layout

In the MicroStation environment, our products provide a set of extended options and functionality beyond those available in stand-alone environment. This additional functionality provides enhanced control over general application settings and options and extends the command set, giving you control over the display of model elements within MicroStation.

It is important to be aware that there are two lists of menu items when running WaterGEMS V8*i* in MicroStation:

1. MicroStation menu (File Edit Element Settings ...) which contains MicroStation commands. The MicroStation menu contains commands which affect the drawing.
2. WaterGEMS V8*i* menu (Project Edit Analysis ...) which contains WaterGEMS V8*i* commands. The WaterGEMS V8*i* menu contains commands which affect the hydraulic analysis.

It is important to be aware of which menu you are using.

Key differences between MicroStation and stand-alone environment include:

- Full element symbol editing functionality is available through the use of custom cells. All elements and graphical decorations (flow arrows, control indicators, etc.) are contained within a WaterGEMS V8*i* .cel file. To do this open the .cel file that's in the WTRG install directory in MSTN (at the first, Open dialog), and then using the File>models you can select each of the WTRG symbols and change them using normal MSTN commands. Then when you create a new dgn and start laying out the WTRG elements, the new symbols will be used.
- The more powerful Selection tools are in the MicroStation select menu.
- Element symbols like junction are circles that are not filled. The user must pick the edge of the circle, not inside the circle to pick a junction.
- The MicroStation background color is found in Workspace>Preferences>View Options. It can also be changed in Settings>Color Tab.
- Zooming and panning are controlled by the MicroStation zooming and panning tools.
- Depending on how MicroStation was set up, a single right click will simply clear the last command, while holding down the right mouse button will bring up the context sensitive menu. There are commands in that menu (e.g. rotate) that are not available in WaterGEMS V8*i* stand alone.

You can control the appearance and destination of all model elements using the Element Levels command under the View menu. For example, you can assign a specific level for all outlets, as well as assign the label and annotation text style to be applied. Element attributes are either defined by the MicroStation Level Manager, using by-level in the attributes toolbox, or by the active attributes. You can change the element attributes using the change element

attributes tool, located in the change attributes toolbox, located on the MicroStation Main menu.

WaterGEMS V8*i* toolbars are turned off by default when you start. They are found under View>Toolbars and they can be turned on. By default they will be floating toolbars but they can be docked wherever the user chooses.

*Note: Any MicroStation tool that deletes the target element (such as **Trim** and **IntelliTrim**) will also remove the connection of that element to WaterGEMS V8*i*. After the WaterGEMS V8*i* connection is removed, the element is no longer a valid wtg link element and will not show properties on the property grid. The element does not have properties because it is not part of the WTRG model. It is as if the user just used MSTN tools to lay out a rectangle in a WTRG dgn. It is just a dgn drawing element but has nothing to do with the water model.*

3.7 Working in AutoCAD

The AutoCAD environment lets you create and model your network directly within your primary drafting environment. This gives you access to all of AutoCAD's drafting and presentation tools, while still enabling you to perform Bentley WaterGEMS V8*i* modeling tasks like editing, solving, and data management. This relationship between Bentley WaterGEMS V8*i* and AutoCAD enables extremely detailed and accurate mapping of model features, and provides the full array of output and presentation features available in AutoCAD. This facility provides the most flexibility and the highest degree of compatibility with other CAD-based applications and drawing data maintained at your organization.

Bentley WaterGEMS V8*i* features support for AutoCAD integration. You can determine if you have purchased AutoCAD functionality for your license of Bentley WaterGEMS V8*i* by using the **Help > About** menu option. Click the **Registration** button to view the feature options that have been purchased with your application license. If AutoCAD support is enabled, then you will be able to run your Bentley WaterGEMS V8*i* application in both AutoCAD and stand-alone environment.

The AutoCAD functionality has been implemented in a way that is the same as the WaterGEMS V8*i* base product. Once you become familiar with the stand-alone environment, you will not have any difficulty using the product in the AutoCAD environment.

Some of the advantages of working in the AutoCAD environment include:

- Layout network links and structures in fully-scaled environment in the same design and drafting environment that you use to develop your engineering plans. You will have access to any other third-party applications that you currently use, along with any custom LISP, ARX, or VBA applications that you have developed.
- Use native AutoCAD insertion snaps to precisely position Bentley WaterGEMS V8*i* elements with respect to other entities in the AutoCAD drawing.
- Use native AutoCAD commands such as ERASE, MOVE, and ROTATE on Bentley WaterGEMS V8*i* model entities with automatic update and synchronization with the model database.
- Control destination layers for model elements and associated label text and annotation, giving you control over styles, line types, and visibility of model elements.

Note: Bentley WaterGEMSV8i supports the 32-bit and 64-bit versions of

AutoCAD 2012, 2013, and 2014 only.

Caution!	*If you previously installed Bentley ProjectWise and turned on AutoCAD integration, you must add the following key to your system registry using the Windows Registry Editor. Before you edit the registry, make a backup copy.* *HKEY_LOCAL_MACHINE\SOFTWARE\Bentley\ProjectWise iDesktop Integration\XX.XX\Configuration\AutoCAD"* *String value name: DoNotChangeCommands* *Value: 'On'* *To access the Registry Editor, click Start > Run, then type regedit. Using the Registry Editor incorrectly can cause serious, system-wide problems that may require you to re-install Windows to correct them. Always make a backup copy of the system registry before modifying it.*

3.8 Working in ArcGIS

Bentley WaterGEMS V8i provides three environments in which to work: Bentley WaterGEMS V8i Stand-Alone Mode, AutoCAD Integrated Mode, and ArcMap Integrated Mode. Each mode provides access to differing functionality—certain capabilities that are available within Bentley WaterGEMS V8i Stand-Alone mode may not be available when working in ArcMap Integrated mode, and vice-versa. In addition, you can use ArcCatalog to perform actions on any Bentley WaterGEMS V8i database. Some of the advantages of working in GIS mode include:

- Full functionality from within the GIS itself, without the need for data import, export, or transformation
- The ability to view and edit multiple scenarios in the same geodatabase
- Minimizes data replication
- GIS custom querying capabilities
- Lets you build models from scratch using practically any existing data source
- Utilizes the powerful reporting and presentation capabilities of GIS

A firm grasp of GIS basics will give you a clearer understanding of how Bentley WaterGEMS V8*i* interacts with GIS software. Click one the following links to learn more:

- ArcGIS Integration
- ArcGIS Applications

3.9 ArcGIS Integration

Bentley WaterGEMS V8*i* features full integration with ESRI's ArcGIS software, including ArcView, ArcEdit, and ArcInfo. The following is a description of the functionality available with each of these packages:

- **ArcView** – ArcView provides the following capabilities:
  - Data Access
  - Mapping
  - Customization
  - Spatial Query

- Simple Feature Editing

  ArcView can edit shapefiles and personal geodatabases that contain simple features such as points, lines, polygons, and static annotation. Rules and relationships cannot be edited with ArcView.

- **ArcEdit** – ArcEdit provides all of the capabilities available with ArcView in addition to the following:
  - Coverage and geodatabase editing

    ArcEdit can edit shapefiles, coverages, personal geodatabases, and multi-user geodatabases.

- **ArcInfo** – ArcInfo provides all of the capabilities available with ArcEdit in addition to the following:
  - Advanced geoprocessing
  - Data conversion
  - ArcInfo Workstation

    ArcInfo can edit shapefiles, coverages, personal geodatabases, and multi-user geodatabases.

3.10 Types of Menus

3.10.1 File Menu

The File menu contains the following commands:

New	Creates a new project. When you select this command, a new untitled project is created.
Open	Opens an existing project. When you select this command, the Open dialog box opens, so you can choose which program to open.
Close	Closes the current project without exiting the program.
Close All	Closes all currently open projects.
Save	Saves the current project.
Save As	Saves the current project under a new project name and/or to a different directory location.
Save All	Saves all currently open projects.
Update Server Copy	Updates the ProjectWise server copy using the current project.
Import	Opens a menu containing the following commands: • WaterGEMS V8*i*/HAMMER **Database** – Opens a Select WaterGEMS V8*i* Database File to Import window, where you can choose the file to import (*.sqlite).

	• **EPANET** – Opens a Select EPANET File to Import window, where you can choose the file to import (*.inp). • **Submodels** – Opens a Select Submodel File to Import window, where you can choose the file to import (*.sqlite).
Export	Opens a menu containing the following commands: • DXF – Exports the current network layout as a DXF drawing. • EPANET – Opens a Select EPANET File to export window where you can choose the file to export (*.inp). • **Submodels** – Exports the current project to a Submodel file (*.sqlite). • **HAMMER 7** – Exports the current project to a WaterGEMS V8*i* input file (.inp). • **Publish i-model** – Opens the Publish to i-model dialog.
Seed	Seed files allow you to save project settings and data as a template (the seed file has an .sts extension). You can then reuse these settings/data while creating new projects using the data from the previously saved seed file. Selecting the Seed command opens a submenu containing the following commands: • **New from Seed**: Allows you to create a new project using the previously saved seed file you specify. • **Save to Seed**: Saves the current project settings and data as a seed file for reuse in future projects.
Page Setup	Opens the Page Setup dialog box where the print settings can be set up.
Print Preview	Opens a submenu containing the following commands: • **Fit to Page** – Opens the Print Preview window, displaying the current view as it will be printed. The view will be zoomed in or out so that the current view fits to a single page of the default page size. • **Scaled** – Opens the Print Preview window, displaying the current view as it will be printed. The view will be scaled so that it matches the user-defined drawing scale (this is defined on the Drawing Tab of the Options dialog: Tools > Options).
Print	Opens a submenu containing the following commands: • **Fit to Page** – Prints the current view. The view will be zoomed in or out so that the current view fits to a single page of the default page size. • **Scaled** – Prints the current view. The view will be scaled so that it matches the user-defined drawing scale (this is defined on the Drawing Tab of the Options dialog: Tools > Options).
Project Properties	Opens the Project Properties dialog box where Title, File Name, Engineer, Company, Date, and Notes can be added.
Recent Files	When the Recent Files Visible option is selected in the Options dialog box, the most recently opened files will appear in the File menu.
Exit	Closes the program.

3.10.2 Edit Menu

The Edit menu contains the following commands:

Undo	Cancels the last data input action on the currently active dialog box. Clicking Undo again cancels the second-to-last data input action, and so on.
Redo	Cancels the last undo command.
Delete	Deletes the currently highlighted element.
Select by Polygon	Selects elements by Polygon.
Select All	Selects all of the elements in the network.
Invert Selection	Selects all of the currently unselected elements in the drawing pane and deselects all of the currently selected elements.
Select by Element	Opens a menu listing all available element types. Select one of the element types from the submenu to select all elements of that type in the model.
Select by Attribute	Opens a menu listing all available attribute types. Select one of the attribute types from the menu and the Query Builder dialog box opens.
Clear Selection	Deselects the currently selected element(s).
Clear Highlight	Removes Network Navigator highlighting for all elements.
Find Element	Finds a specific element by entering the element's label.

3.10.3 Analysis Menu

The Analysis menu contains the following commands:

Scenarios	Opens the Scenario Manager, which allows you to create, view, and manage project scenarios.
Alternatives	Opens the Alternative Manager, which allows you to create, view, and manage alternatives.
Calculation Options	Opens the Calculation Options Manager, which allows you to create, view, and manage calculation settings for the project.
Post Calculation Processor	Opens the Post Calculation Processor dialog.
Totalizing Flow Meters	Opens the Totalizing Flow Meters manager where you can create new meters.
Hydrant Flow Curves	Opens the Hydrant Flow Curves dialog box, which allows you to view, edit, and create hydrant flow definitions.
System Head Curves	Opens the System Head Curves manager.
Energy Costs	Opens the Scenario Energy Cost Manager, where you can view and compute energy costs.
Darwin	Opens the Darwin Calibrator, where you can create, edit, and run

Calibrator	calibration studies.
Darwin Designer	Opens the Darwin Designer, where you can create, edit, and run designer studies and design runs.
Darwin Scheduler	Opens the Darwin Scheduler, where you can create, edit, and run scheduler studies and design runs.
Criticality	Opens the Segmentation and Criticality Manager, where you can create new criticality scenarios.
Pressure Zone	Opens the Pressure Zone manager, where you can identify elements that are located in a pressure zone based on the boundaries of the zone.
Time Browser	Opens the Time Browser dialog box, where you can manipulate the currently displayed time step and animate the drawing pane.
Fire Flow Results Browser	Opens the Fire Flow Results Browser, which allows you to quickly jump to fire flow nodes and display the results of fire flow analysis at the highlighted node.
Flushing Results Browser	Opens the Flushing Results Browser, allowing you to display the results of the flushing analysis at various locations.
Calculation Summary	Opens the Calculation Summary to view results.
User Notifications	Opens User Notifications, allowing you to view warnings and errors uncovered by the validation process.
Validate	Runs a diagnostic check on the network data to alert you to possible problems that may be encountered during calculation. This is the manual validation command, and it checks for input data errors. It differs in this respect from the automatic validation that WaterGEMS V8*i* runs when the compute command is initiated, which checks for network connectivity errors as well as many other things beyond what the manual validation checks.
Compute	Calculates the network. Prior to calculating, an automatic validation routine is triggered, which checks the model for network connectivity errors and performs other validation.

3.10.4 Components Menu

The Components menu contains the following commands:

Controls	Opens the Controls manager, where you can set controls, conditions, actions, and logical control sets.
Zones	Opens the Zones manager, where you can create, edit, duplicate, or delete zones.
Patterns	Opens the Patterns manager, where you can create and edit patterns.
Pressure Dependent Demand Functions	Opens the Pressure Dependent Demand Functions manager, where you can create and edit pressure dependent demands.
Unit Demands	Opens the Unit Demands manager, where you can create and edit unit demands based on area, count and population.
Pump Definitions	Opens the Pump Definitions manager, where you can create and edit pump definitions.
Minor Loss Coefficients	Opens the Minor Loss Coefficients Manager dialog.
GPV Headloss Curves	Opens the GPV Headloss Curves manager, where you can create and edit headloss curves for General Purpose Valves.
Constituents	Opens the Constituents manager, where you can create, edit, duplicate, or delete constituents.
Valve Characteristics	Opens the Valve Characteristics dialog.
Air Flow Curves	Opens the Air Flow Curves dialog.
Time Series Field Data	Opens the Time Series Field Data dialog.
Engineering Libraries	Opens the Engineering Libraries Manager.

3.10.5 View Menu

The View menu contains the following commands:

Element Symbology	Opens the Element Symbology Manager, which allows you to create, view, and manage annotation and color-coding in your project.
Background Layers	Opens the Background Layers Manager, which allows you to create, view, and manage the background layers associated with the project.
Network Navigator	Opens the Network Navigator.
Selection Sets	Opens the Selection Sets Manager, which allows you to create, view, and manage selection sets associated with the project.
Queries	Opens the Query Manager, where you can create SQL expressions for use with selection sets and FlexTables.
Prototypes	Opens the Prototypes Manager, where you can enter default values for elements in your model. Prototypes can reduce data entry requirements if a group of network elements share common data.
FlexTables	Opens the FlexTables Manager, where you can create, view, and

	manage the tabular reports for the project.
Graphs	Opens the Graph Manager, where you can create, view, and manage graphs for the project.
Profiles	Opens the Profile Manager, where you can create, view, and manage the profiles for the project.
Contours	Opens the Contours manager where you can create and edit contour definitions.
Named Views	Opens the Named Views manager where you can create, edit, and use Named Views.
Aerial View	Opens the Aerial View navigation window.
Properties	Turns the Properties Editor display on or off.
Property Grid Customizations	Opens the Property Grid Customizations Manager.
Auto-Refresh	Turns automatic updates to the main window view on or off whenever changes are made to the Bentley WaterGEMS V8*i* datastore. When selected, a check mark indicates that automatic updates are turned on.
Refresh Drawing	Updates the main window view according to the latest information contained in the Bentley WaterGEMS V8*i* datastore.
Zoom	Opens a menu containing the following commands: • **Zoom Extents**—Sets the view so that the entire network is visible in the drawing pane. • **Zoom Window**—Activates the manual zoom tool, which lets you specify a portion of the drawing to enlarge. • **Zoom In**—Enlarges the size of the model in the drawing pane. • **Zoom Out**—Reduces the size of the model in the drawing pane. • **Zoom Realtime**—Enables the realtime zoom tool, which allows you to zoom in and out by moving the mouse while holding down the left mouse button. • **Zoom Center**—Opens the Zoom Center dialog box, which allows you to enter drawing coordinates that will be centered in the drawing pane. • **Zoom to Selection**—Enables you to zoom to specific elements in the drawing. You must select the elements to zoom to before you select the tool. • **Zoom Previous**—Resets the zoom level to the last setting. • **Zoom Next**—Resets the zoom level to the setting that was active before a Zoom Previous command was executed.
Pan	Activates the Pan tool, which allows you to move the model within the drawing pane. When you select this command, the cursor changes to a hand, indicating that you can click and hold the left mouse button and move the mouse to move the drawing.
Toolbars	Opens a menu that lists each of the available toolbars. Select one of the toolbars in the menu to turn that toolbar on or off.
Reset Workspace	Resets the Bentley WaterGEMS V8*i* workspace so that the dockable managers appear in their default factory-set positions.

3.10.6 Tools Menu

The Tools menu contains the following commands:

Active Topology Selection	Opens a Select dialog to select elements in the drawing to make them Inactive or Active.
ModelBuilder	Opens the ModelBuilder Connections Manager, where you can create, edit, and manage ModelBuilder connections to be used in the model-building/model-synchronizing process.
TRex	Opens the TRex wizard, where you can assign elevation to model nodes using data from outside sources.
SCADAConnect	Opens the SCADAConnect manager, where you can add or edit SCADA connections.
Skelebrator Skeletonizer	Opens the Skelebrator manager, where you can define and perform skeletonization operations.
LoadBuilder	Opens the LoadBuilder manager, where you can assign demands to model nodes using data from outside sources.
Thiessen Polygon	Opens the Wizard used to create Thiessen polygons for use with LoadBuilder.
Demand Control Center	Opens the Demand Control Center manager, where you can add new demands, delete existing demands, or modify existing demands.
Unit Demand Control Center	Opens the Unit Demand Control Center manager, where you can add new unit demands, delete existing unit demands, or modify existing unit demands.
Scenario Comparison	The scenario comparison tool enables you to compare input values between any two scenarios to identify differences quickly.
Hyperlinks	Associate external files, such as pictures or movie files, with elements in the model.
User Data Extensions	Opens the User Data Extension dialog box, which allows you to add and define custom data fields. For example, you can add new fields such as the pipe installation date.
Assign Isolation Valves to Pipes	Opens the Assign Isolation Valves to Pipes, where you can find and assign isolation valves to their closest pipes according to user-defined tolerances.
Batch Pipe Split	Opens the Batch Pipe Split dialog.
Batch Morph	Opens the Batch Morph dialog.
Database Utilities	Opens a menu containing the following commands: • **Compact Database**—When you delete data from a Bentley WaterGEMS V8*i* project, such as elements or alternatives, the database store that Bentley WaterGEMS V8*i* uses can become fragmented, causing unnecessarily large data files, which impact performance substantially. Compacting the database eliminates the empty data records, thereby defragmenting the datastore and improving the performance of the file. *Note: Every tenth time a file is saved, Bentley WaterGEMS*

	V8i will automatically prompt you to compact the database. If you open a file without saving it, the count does not go up. If you open and save a file multiple times in the same session, the count only goes up on the first save. If you open, save, and close the file, the count goes up. Click Yes to compact the database, or no to close the prompt dialog box without compacting. Since compacting the database can take time, especially for larger models, you may want to postpone the compact procedure until a later time. You can modify how Bentley WaterGEMS V8i compacts the database in the Options dialog box. • **Synchronize Drawing**—Synchronizes the current model drawing with the project database. • **Update Database Cache**—Updates the current model to reflect any changes made in the database. • **Update Results from Project Directory**—This command copies the model result files (if any) from the project directory (the directory where the project .sqlite file is saved) to the custom result file directory. The custom result directory is specified in Tools>Options>Project tab. This allows you to make a copy of the results that may exist in the model's save directory and replace the current results being worked on with them. • **Copy Results to Project Directory**—This command copies the result files that are currently being used by the model to the project directory (where the project .sqlite is stored).
Layout	Opens a menu that lists each of the available element types. Select one of the element types to place that element in your model.
External Tools	Run an existing external tool or create a new one by opening up the External Tools manager.
Options	Opens the Options dialog box, which allows you to change Global settings, Drawing, Units, Labeling, and ProjectWise.

3.10.7 Report Menu

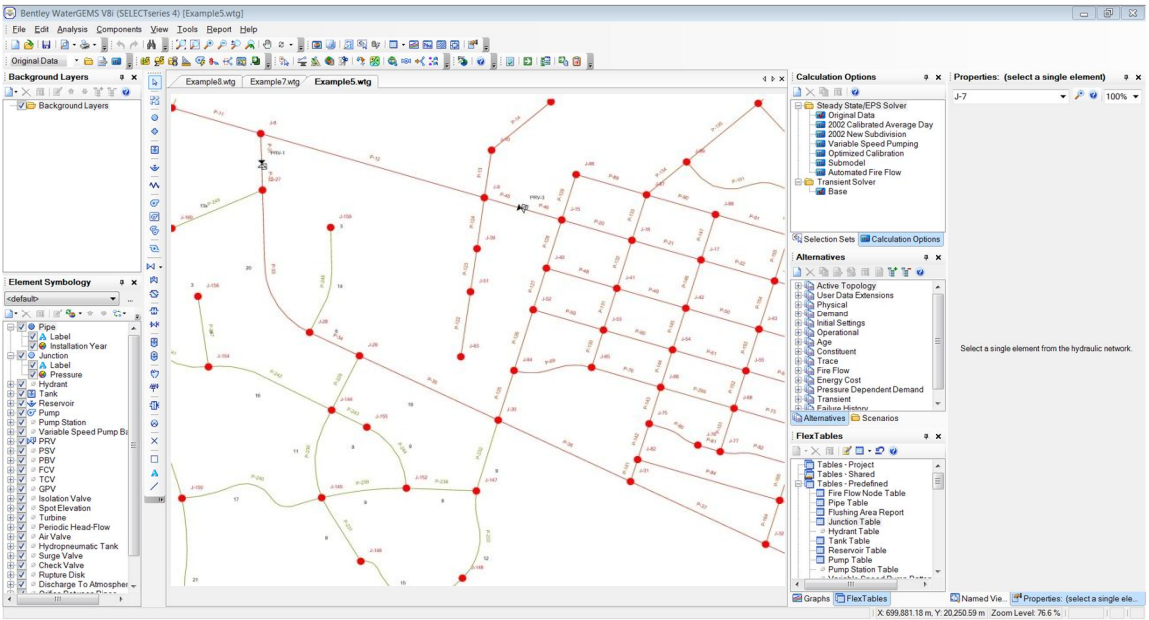
The Report menu contains the following commands:

Element Tables	Opens a menu that allows you to display FlexTables for any link or node element. These predefined FlexTables contain most of the input data and results for each instance of the selected element in the model.
Scenario Summary	Opens the Scenario Summary Report.
Project Inventory	Opens the Project Inventory Report, which contains the number of each of the various element types that are in the network.
Pressure Pipe Inventory	Opens the Pressure Pipe Inventory report.
Report Options	Opens the Report Options box where you can set Headers and Footers for the predefined reports.

3.10.8 Help Menu

The Help menu contains the following commands:

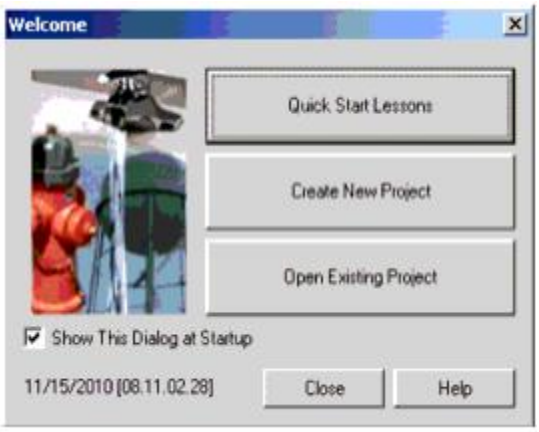
Bentley WaterGEMS V8i Help	Opens the online help Table of Contents.
Quick Start Lessons	Opens the online help to the Quick Start Lessons Overview topic.
Welcome Dialog	Opens the Welcome dialog box.
Check for SELECT Updates	Opens your Web browser to the Bentley Web site, where you can check for Bentley WaterGEMS V8*i* updates.
Bentley Institute Training	Opens your browser to the Bentley Institute Training web site.
Bentley Professional Services	Opens your browser to the Bentley Professional Services web site.
Bentley SELECT Support	Opens your browser to SELECTservices area of the Bentley web site.
Bentley Communities	Opens your browser to the BentleyCommunities section of the website.
Bentley.com	Opens the home page on the Bentley web site.
About Bentley WaterGEMS V8i	Opens the About Bentley Bentley WaterGEMS V8*i* dialog box, which displays copyright information about the product, registration information, and the current version number of the release.



3.11 Types of WaterGEMS

3.11.1 Starting a Project

When you first start Bentley WaterGEMS V8i, the Welcome dialog box opens. The Welcome dialog box contains the following controls:



Quick Start Lessons	Opens the online help to the Quick Start Lessons Overview topic.
Create New Project	Creates a new WaterGEMS V8i project. When you click this button, an untitled Bentley WaterGEMS V8i project is created.
Open Existing Project	Opens an existing project. When you click this button, a Windows browse dialog box opens allowing you to browse to the project to be opened. If you have ProjectWise installed and integrated with WaterGEMS V8i, you are prompted to log into a ProjectWise datasource if you are not already logged in.
Show This Dialog at Start	When selected, the Welcome dialog box opens whenever you start Bentley WaterGEMSV8i . Turn off this box if you do not want the Welcome dialog box to open whenever you start Bentley WaterGEMSV8i .

To Access the Welcome Dialog during Program Operation:
Click the Help menu and select the Welcome Dialog command.

To Disable the Automatic Display of the Welcome Dialog upon Startup:
In the Welcome dialog, turn off the box labeled Show This Dialog at Start.

To Enable the Automatic Display of the Welcome Dialog upon Startup:
In the Welcome dialog, turn on the box labeled Show This Dialog at Start.

3.12 Bentley WaterGEMS V8i Projects

All data for a model are stored in WaterGEMS V8i as a project. WaterGEMS V8i project files have the file name extension .wtg. You can assign a title, date, notes and other identifying information about each project using the Project Properties dialog box. You can have up to five WaterGEMS V8i projects open at one time.

To Start a New Project
To start a new project, choose File > New or press <Ctrl+N>. An untitled project is opened in the drawing pane.

To Open an Existing Project
To open an existing project, choose File >Open or press <Ctrl+O>. A dialog box opens allowing you to browse for the project you want to open.

To Switch Between Multiple Projects
To switch between multiple open projects, select the appropriate tab at the top of the drawing pane. The file name of the project is displayed on the tab.

Files associated with WaterGEMS projects
Main file types:

- *.wtg – is the basic WaterGEMS display settings (e.g. color coding, annotation)
- *.wtg.sqlite – is the model database file
- *.wtg.dwh – is the standalone drawing file
- *.out - primary output file from hydraulic and water quality analyses
- *.bak - backup files of the model files

Other File types:

- *.cri - results of criticality analysis
- *.dgn - drawing file for MicroStation platform
- *.dwg - drawing file for AutoCAD platform
- *.dwh - drawing file for stand-alone platform
- *.sqlite - access database file for ArcGIS platform (may also be a *.mdb file)
- *.nrg - results of energy calculations
- *.osm - outage segmentation results
- *.out.fl - output file from flushing analysis
- *.rpc - report file from hydraulic analysis with user notifications
- *.seg - results of segmentation analysis
- *.xml - xml files, generally libraries, window and other settings. Some modules like ModelBuilder also use .xml files to store settings independent of the main model.

3.13 Setting Project Properties

The Project Properties dialog box allows you to enter project-specific information to help identify the project. Project properties are stored with the project.

The dialog box contains the following text fields and controls:

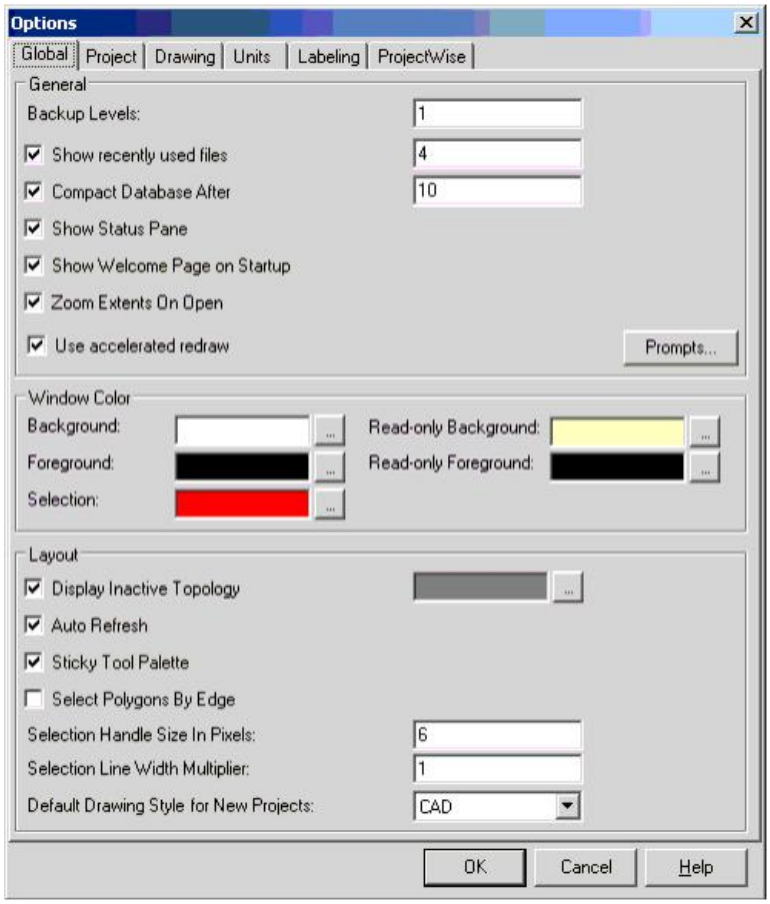
Title	Enter a title for the project.
File Name	Displays the file name for the current project. If you have not saved the project yet, the file name is listed as "Untitledx.wtg.", where x is a number between 1 and 5 chosen by the program based on the number of untitled projects that are currently open.
Engineer	Enter the name of the project engineer.
Company	Enter the name of your company.
Date	Click this field to display a calendar, which is used to set a date for the project.
Notes	Enter additional information about the project.

To set project properties:

1. Choose File > Project Properties and the Project Properties dialog box opens.
2. Enter the information in the Project Properties dialog box and click OK.

3.14 Setting Options

You can change global settings for WaterGEMS V8i in the Options dialog box. Choose Tools > Options. The Options dialog box contains different tabs where you can change settings.



Global Tab

The Global tab changes general program settings for the WaterGEMS V8*i* stand-alone editor, including whether or not to display the status pane, as well as window color and layout settings.

The Global tab contains the following controls:

General Settings

Backup Levels Indicates the number of backup copies that are retained when a project is saved. The default value is 1.

Note: The higher this number, the more .BAK files (backup files) are created, thereby using more hard disk space on your computer.

Show Recently Used Files When selected, activates the recently opened files display at the bottom of the File menu. This check box is turned on by default. The number of recently used files that are displayed depends on the number specified here.

Compact Database After	When this box is checked the WaterGEMS V8*i* database is automatically compacted when you choose File > Open after the file has been opened the number of times specified here.
Show Status Pane	When turned on, activates the Status Pane display at the bottom of the WaterGEMS V8*i* stand-alone editor. This check box is turned on by default.
Show Welcome Page on Startup	When turned on, activates the Welcome dialog that opens when you first start WaterGEMS V8*i*. This check box is turned on by default.
Zoom Extents On Open	When turned on, a Zoom Extents is performed automatically in the drawing pane.
Use accelerated redraw	Some video cards use "triple buffering", which we do not support at this time. If you see anomalies in the drawing (such as trails being left behind from the selection rectangle), then you can shut this option off to attempt to fix the problem. However, when this option is off, you could see some performance degradation in the drawing.
Prompts	Opens the Stored Prompt Responses dialog, which allows you to change the behavior of the default prompts (messages that appear allowing you to confirm or cancel certain operations).

Window Color

Background Color	Displays the color that is currently assigned to the drawing pane background. You can change the color by clicking the ellipsis (...) to open the Color dialog box.
Foreground Color	Displays the color that is currently assigned to elements and labels in the drawing pane. You can change the color by clicking the ellipsis (...) to open the Color dialog box.
Read Only Background Color	Displays the color that is currently assigned to read-only data field backgrounds. You can change the color by clicking the ellipsis (...) to open the Color dialog box.
Read Only Foreground Color	Displays the color that is currently assigned to read-only data field text. You can change the color by clicking the ellipsis (...) to open the Color dialog box.
Selection Color	Displays the color that is currently applied to highlighted elements in the drawing pane. You can change the color by clicking the ellipsis (...) to open the Color dialog box.

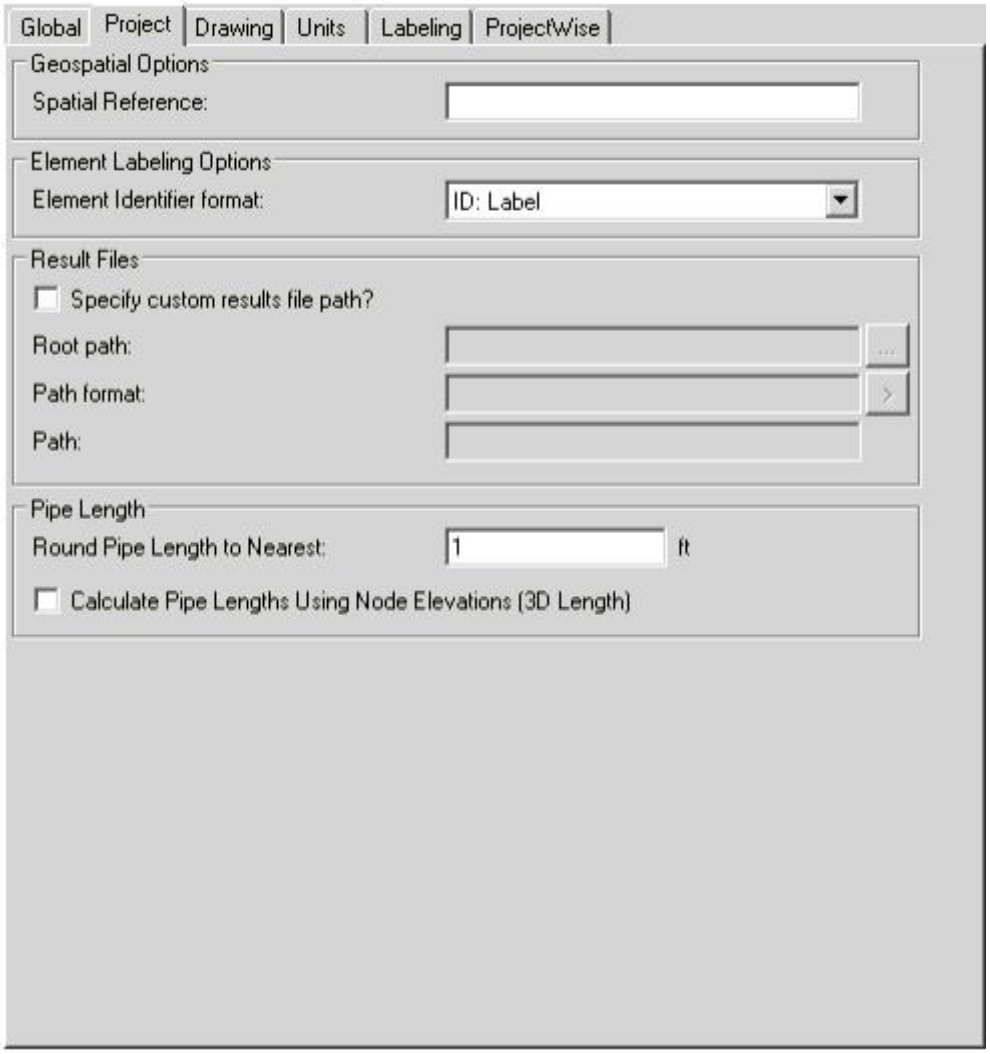
Layout

Display Inactive Topology	When turned on, activates the display of inactive elements in the drawing pane in the color defined in Inactive Topology Line Color. When turned off, inactive elements will not be visible in the drawing pane. This check box is turned on by default.
Inactive Topology Line Color	Displays the color currently assigned to inactive elements. You can change the color by clicking the ellipsis (...) to open the Color dialog box.

Auto Refresh	Activates Auto Refresh. When Auto Refresh is turned on, the drawing pane automatically updates whenever changes are made to the WaterGEMS V8*i* datastore. This check box is turned off by default.
Sticky Tool Palette	When turned on, activates the Sticky Tools feature. When Sticky Tools is turned on, the drawing pane cursor does not reset to the Select tool after you create a node or finish a pipe run in your model, allowing you to continue dropping new elements into the drawing without re-selecting the tool. When Sticky Tools is turned off, the drawing pane cursor resets to the Select tool after you create a node. This check box is selected by default.
Select Polygons By Edge	When this box is checked, polygon elements (catchments) can only be selected in the drawing pane by clicking on their bordering line, in other words you cannot select polygons by clicking their interior when this option is turned on.
Selection Handle Size In Pixels	Specifies, in pixels, the size of the handles that appear on selected elements. Enter a number from 1 to 10.
Selection Line Width Multiplier	Increases or decreases the line width of currently selected link elements by the factor indicated. For example, a multiplier of 2 would result in the width of a selected link being doubled.
Default Drawing Style	Allows you to select GIS or CAD drawing styles. Under GIS style, the size of element symbols in the drawing pane will remain the same regardless of zoom level. Under CAD style, element symbols will appear larger or smaller depending on zoom level.

Project Tab

This tab contains miscellaneous settings. You can set pipe length calculation, spatial reference, label display, and results file options in this tab.



The Project tab contains the following controls:

Geospatial Options

Spatial Reference Used for integration with Projectwise. Can leave the field blank if there is no spatial information.

Element Identifier Options

Element Identifier Format Specifies the format in which reference fields are used. Reference fields are fields that link to another element or support object (pump definitions, patterns, controls, zones, etc.).

Result Files

Specify Custom Results File Path? When checked, allows you to edit the results file path and format by enabling the other controls in this section.

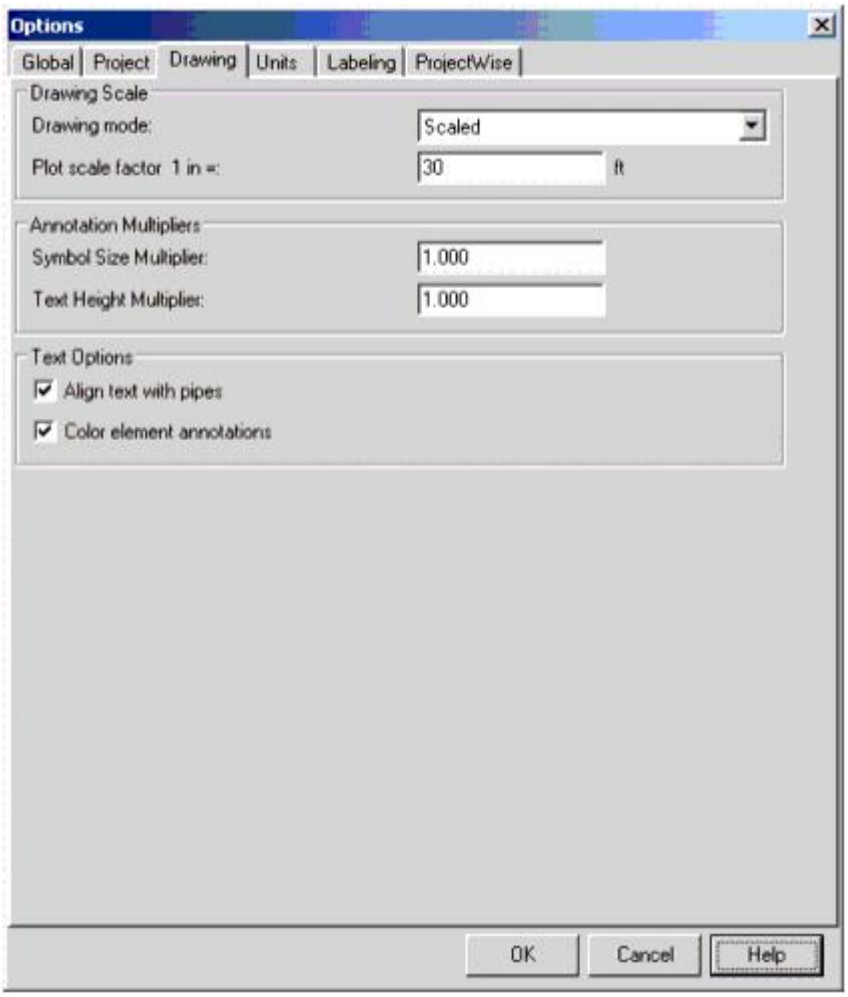
Root Path Allows you to specify the root path where results files are stored. You can type the path manually or choose the path from a Browse dialog by clicking the ellipsis (...) button.

Path Format Allows you to specify the complete path that you wish to use for storing your result files for the current project. You can type the

	path manually and/or use predefined attributes from the menu accessed with the [>] button. One of the predefined choices is the Root Path. It is recommended that you start building your Path Format with this Root Path choice. Then optionally extend this path with the other predefined choices.
Path	Displays a dynamically updated view of the custom result file path based on the settings in the Root Path and Path Format fields
	Pipe Length
Round Pipe Length to Nearest	The program will round to the nearest unit specified in this field when calculating scaled pipe length
Calculate Pipe Lengths Using Node Elevations (3D Length)	When checked, includes differences in Z (elevation) between pipe ends when calculating pipe length.

Drawing Tab

This tab contains drawing layout and display settings. You can set the scale that you want to use as the finished drawing scale for the plan view output. Drawing scale is based upon engineering judgment and the destination sheet sizes to be used in the final presentation.



The Drawing tab contains the following controls:

Drawing Scale

Drawing Mode Selects either Scaled or Schematic mode for models in the drawing pane.

Horizontal Scale Factor 1 in. =: Controls the scale of the plan view.

Annotation Multipliers

Symbol Size Mulitplier Increases or decreases the size of your symbols by the factor indicated. For example, a multiplier of 2 would result in the symbol size being doubled. The program selects a default symbol height that corresponds to 4.0 ft. (approximately 1.2 m) in actual-world units, regardless of scale.

Text Height Multiplier Increases or decreases the default size of the text associated with element labeling by the factor indicated. The program automatically selects a default text height that displays at approximately 2.5 mm (in) high at the user-defined drawing scale. A scale of 1.0 mm = 0.5 for example, results in a text height of approximately 1.25 m. Likew a 1 in. = 40 ft. scale equates to a text height of around 4.0 ft.

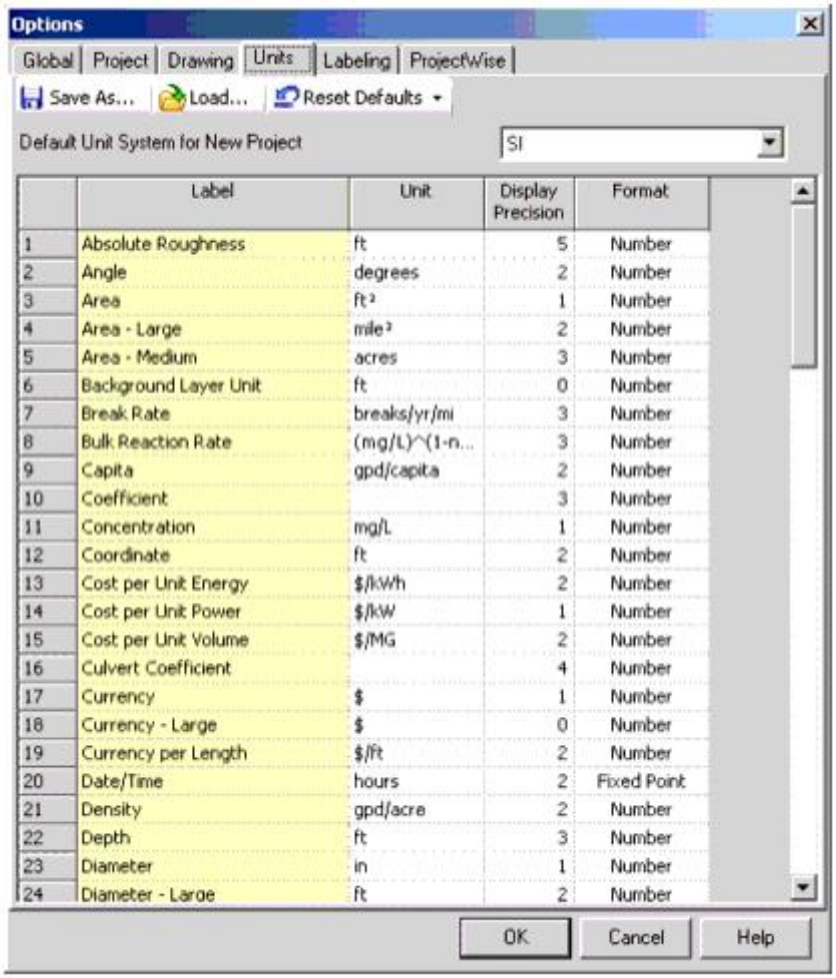
Text Options

Align Text with Pipes Turns text alignment on and off. When it is turned on, labels are aligned to their associated pipes. When it is turned off, labels are displayed horizontally near the center of the associated pipe.

Color Element Annotations When this box is checked, color coding settings are applied to the element annotation.

Units Tab

The Units tab modifies the unit settings for the current project.



The Units tab contains the following controls:

Save As Saves the current unit settings as a separate .xml file. This file allows you to reuse your Units settings in another project. When the button is clicked, a Windows Save As dialog box opens, allowing you to enter a name and specify the directory location of the .xml file.

Load Loads a previously created Units project .xml file, thereby transferring the unit and format settings that were defined in the previous project. When the button is clicked, a Windows Load dialog box opens, allowing you to browse to the location of the desired .xml file.

Reset Defaults - SI Resets the unit and formatting settings to the original factory defaults for the System International (Metric) system.

Reset Defaults - US Resets the unit and formatting settings to the original factory defaults for the Imperial (U.S.) system.

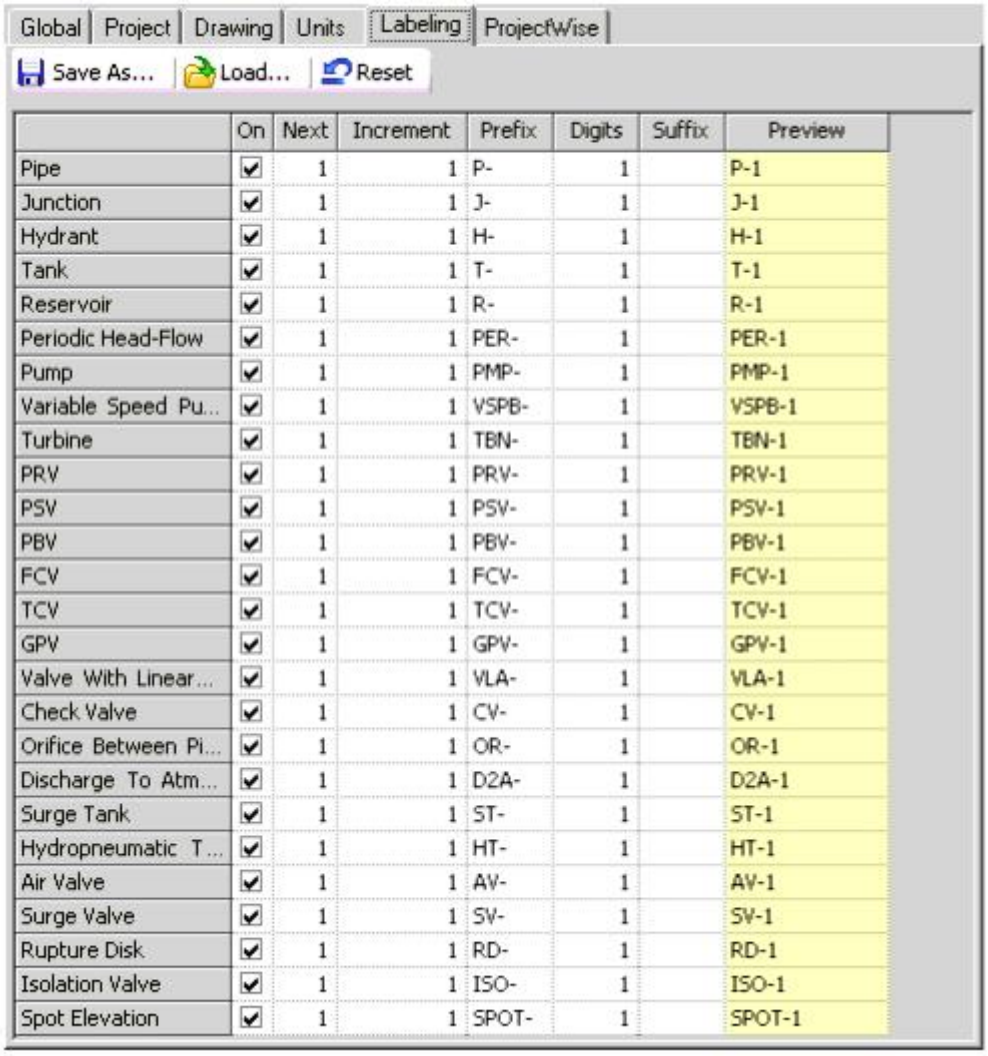
Default Unit System for New Project Specifies the unit system that is used globally across the project. Note that you can locally change any number of attributes to the unit system other than the ones specified here.

Units Table The units table contains the following columns:

- **Label**—Displays the parameter measured by the unit.
- **Unit**—Displays the type of measurement. To change the unit of an attribute type, click the choice list and click the unit you want. This option also allows you to use both U.S. customary and SI units in the same worksheet.
- **Display Precision**—Sets the rounding of numbers and number of digits displayed after the decimal point. Enter a number from 0 to 15 to indicate the number of digits after the decimal point.
- **Format Menu**—Selects the display format used by the current field. Choices include:
  - **Scientific**—Converts the entered value to a string of the form "-d.ddd...E+ddd" or "-d.ddd...e+ddd", where each 'd' indicates a digit (0-9). The string starts with a minus sign if the number is negative.
  - **Fixed Point**—Abides by the display precision setting and automatically enters zeros after the decimal place to do so. With a display precision of 3, an entered value of 3.5 displays as 3.500.
  - **General**—Truncates any zeros after the decimal point, regardless of the display precision value. With a display precision of 3, the value that would appear as 5.200 in Fixed Point format displays as 5.2 when using General format. The number is also rounded. So, an entered value of 5.35 displays as 5.4, regardless of the display precision.
  - **Number**—Converts the entered value to a string of the form "-d,ddd,ddd.ddd...", where each 'd' indicates a digit (0-9). The string starts with a minus sign if the number is negative. Thousand separators are inserted between each group of three digits to the left of the decimal point.

Labeling Tab

The Element Labeling tab is used to specify the automatic numbering format of new elements as they are added to the network. You can save your settings to an .xml file for later use.



	On	Next	Increment	Prefix	Digits	Suffix	Preview
Pipe	☑	1	1	P-	1		P-1
Junction	☑	1	1	J-	1		J-1
Hydrant	☑	1	1	H-	1		H-1
Tank	☑	1	1	T-	1		T-1
Reservoir	☑	1	1	R-	1		R-1
Periodic Head-Flow	☑	1	1	PER-	1		PER-1
Pump	☑	1	1	PMP-	1		PMP-1
Variable Speed Pu...	☑	1	1	VSPB-	1		VSPB-1
Turbine	☑	1	1	TBN-	1		TBN-1
PRV	☑	1	1	PRV-	1		PRV-1
PSV	☑	1	1	PSV-	1		PSV-1
PBV	☑	1	1	PBV-	1		PBV-1
FCV	☑	1	1	FCV-	1		FCV-1
TCV	☑	1	1	TCV-	1		TCV-1
GPV	☑	1	1	GPV-	1		GPV-1
Valve With Linear...	☑	1	1	VLA-	1		VLA-1
Check Valve	☑	1	1	CV-	1		CV-1
Orifice Between Pi...	☑	1	1	OR-	1		OR-1
Discharge To Atm...	☑	1	1	D2A-	1		D2A-1
Surge Tank	☑	1	1	ST-	1		ST-1
Hydropneumatic T...	☑	1	1	HT-	1		HT-1
Air Valve	☑	1	1	AV-	1		AV-1
Surge Valve	☑	1	1	SV-	1		SV-1
Rupture Disk	☑	1	1	RD-	1		RD-1
Isolation Valve	☑	1	1	ISO-	1		ISO-1
Spot Elevation	☑	1	1	SPOT-	1		SPOT-1

The Element Labeling tab contains the following controls:

Save As Saves your element labeling settings to an element label project file, which is an. xml file.

Load Opens an existing element label project file.

Reset Assigns the correct Next value for all elements based on the elements currently in the drawing and the user-defined values set in the Increment, Prefix, Digits, and Suffix fields of the Labeling table.

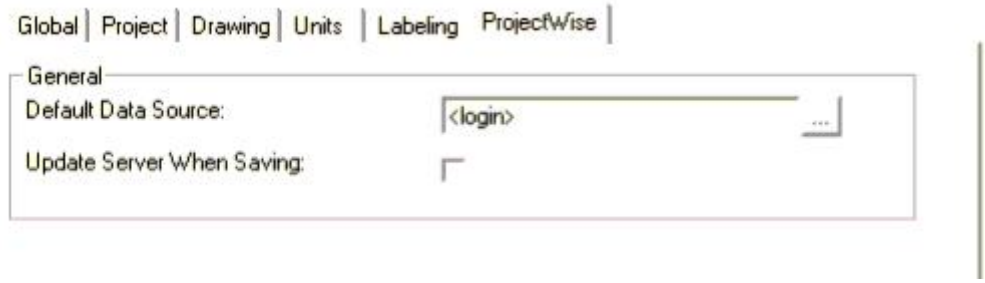
Labeling Table The labeling table contains the following columns:

- **Element**—Shows the type of element to which the label applies.
- **On**—Turns automatic element labeling on and off for the associated element type.

- **Next**—Type the integer you want to use as the starting value for the ID number portion of the label. Bentley WaterGEMS V8*i* generates labels beginning with this number and chooses the first available unique label.
- **Increment**—Type the integer that is added to the ID number after each element is created to yield the number for the next element.
- **Prefix**—Type the letters or numbers that appear in front of the ID number for the elements in your network.
- **Digits**—Type the minimum number of digits that the ID number has. For instance, 1, 10, and 100 with a digit setting of two would be 01, 10, and 100.
- **Suffix**—Type the letters or numbers that appear after the ID number for the elements in your network.
- **Preview**—Displays what the label looks like based on the information you have entered in the previous fields.

ProjectWise Tab

The ProjectWise tab contains options for using WaterGEMS V8*i* with ProjectWise.



This tab contains the following controls:

Default Datasource Displays the current ProjectWise datasource. If you have not yet logged into a datasource, this field will display <login>. To change the datasource, click the **Ellipses (...)** to open the Change Datasource dialog box. If you click **Cancel** after you have changed the default datasource, the new default datasource is retained.

Update server on Save When this is turned on, any time you save your WaterGEMS V8*i* project locally using the File > Save menu command, the files on your ProjectWise server will also be updated and all changes to the files will immediately become visible to other ProjectWise users. This option is turned off by default.

Note: This option, when turned on, can significantly affect performance, especially for large, complex projects.

Note: These settings affect ProjectWise users only.

3.15 Prototypes

Prototypes allow you to enter default values for elements in your network. These values are used while laying out the network. Prototypes can reduce data entry requirements dramatically if a group of network elements share common data.
For example, if a section of the network contains all 12-inch pipes, use the Prototype manager to set the Pipe Diameter field to 12 inches. When you create a new pipe in your model, its diameter attribute will default to 12 inches.

You can create prototypes in either of the following ways:

- From the Prototypes manager: The Prototypes manager consists of a toolbar and a list pane, which displays all of the elements available in WaterGEMS V8i.
- From the Drawing Pane: Right-click an element to use the settings and attributes of that element as the current prototype.

Note: *Changes to the prototypes are not retroactive and will not affect any elements created prior to the change.*

If a section of your system has distinctly different characteristics than the rest of the system, adjust your prototypes before laying out that section. This will save time when you edit the properties later.

To open the Prototypes manager
Choose View > Prototypes

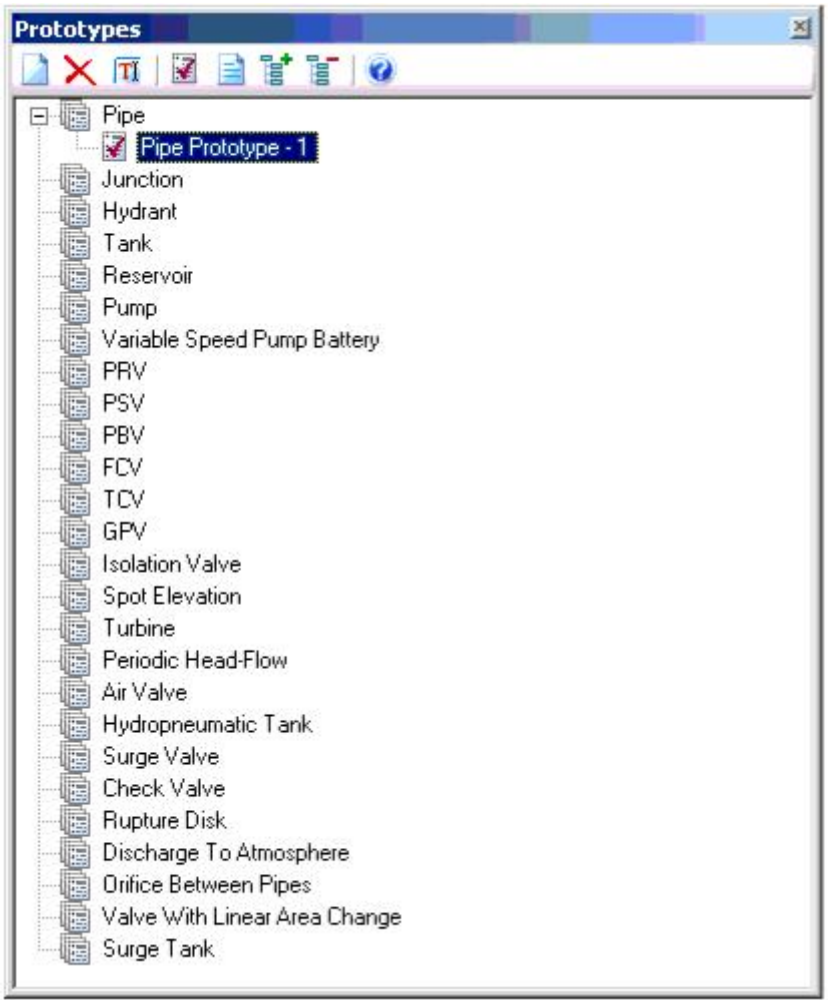
or

Press <Ctrl+6>

or

Click the Prototypes icon  from the View toolbar.

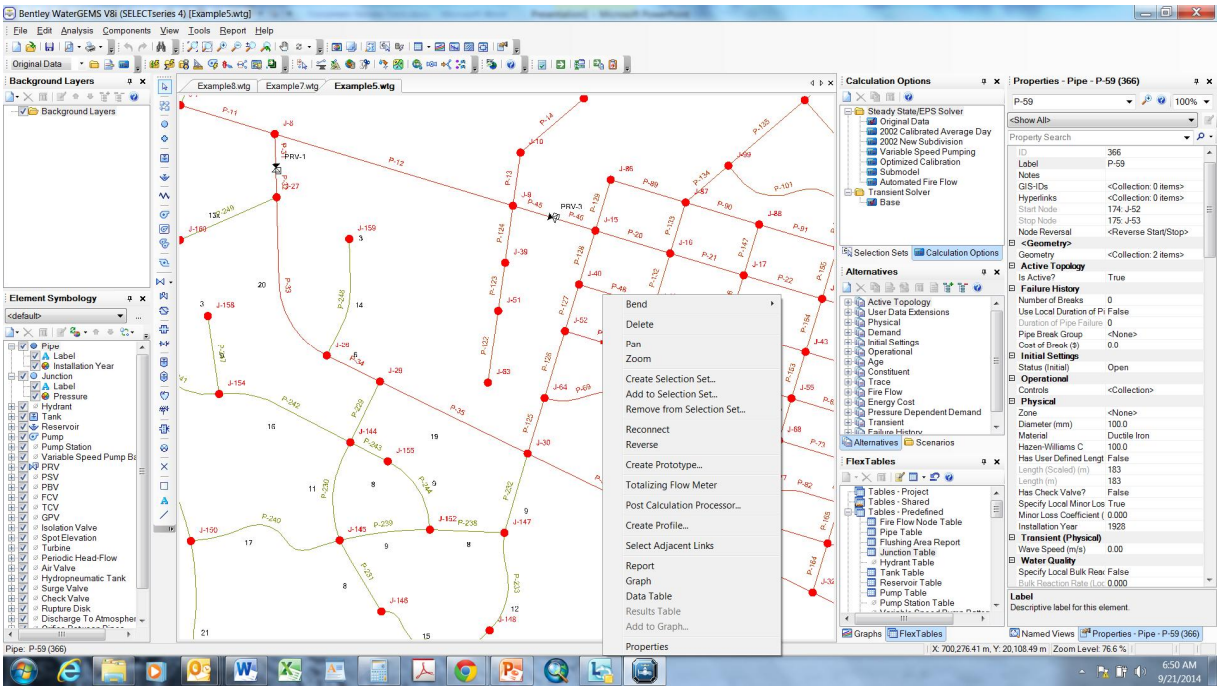
The Prototypes manager then opens.



The list of elements in the Prototypes manager list pane is expandable and collapsible, once you have created additional prototypes. Click on the Plus sign to expand an element and see its associated prototypes. Click on the Minus sign to collapse the element.

Each element in the list pane contains a default prototype; you cannot edit this default prototype. The default prototypes contain common values for each element type; if you add elements to your model without creating new prototypes, the data values in the default prototypes appear in the Property Editor for that element type.

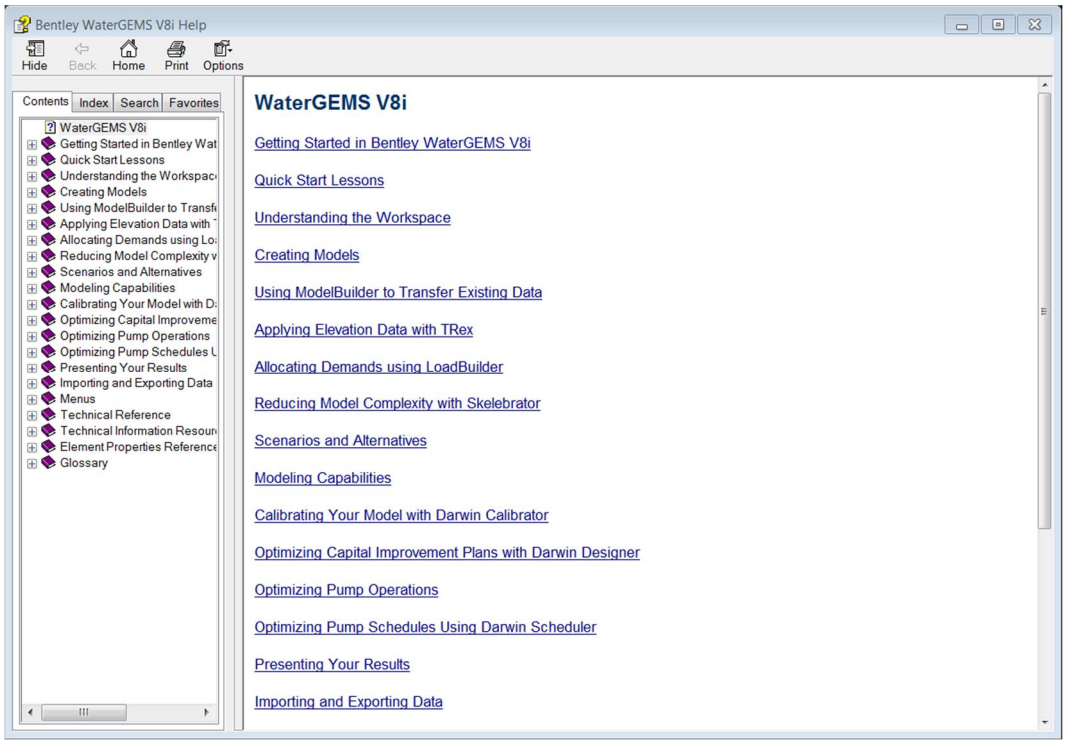
Also you can create a new prototype from current selection.



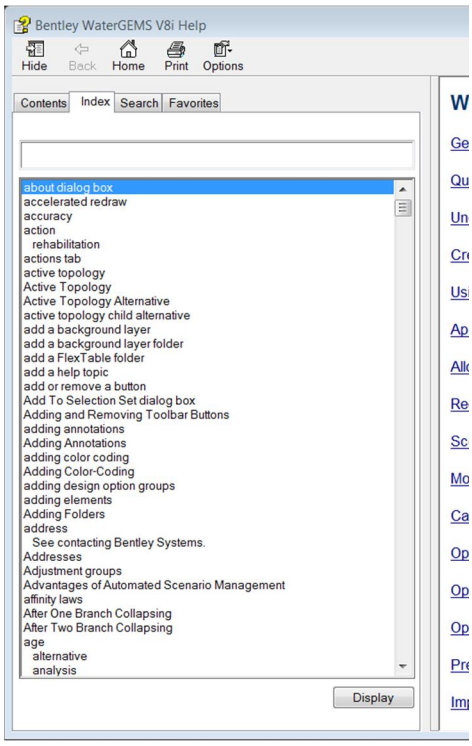
3.16 Help Menu

The Help menu contains the following commands:

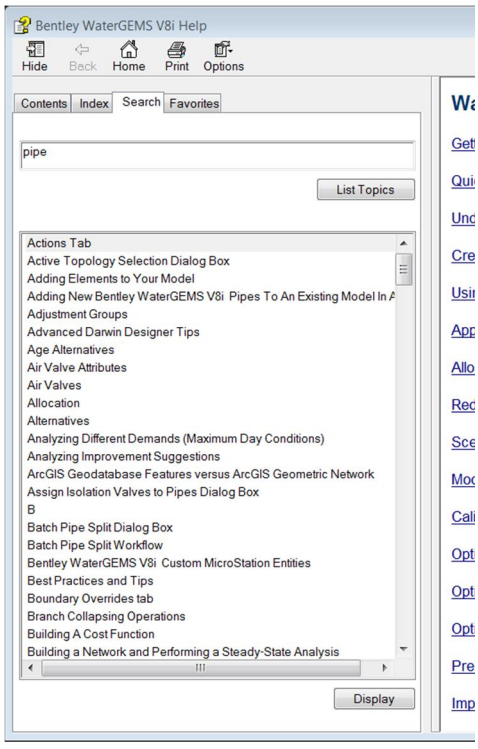
Bentley WaterGEMS V8i Help	Opens the online help Table of Contents.
Quick Start Lessons	Opens the online help to the Quick Start Lessons Overview topic.
Welcome Dialog	Opens the Welcome dialog box.
Check for SELECT Updates	Opens your Web browser to the Bentley Web site, where you can check for Bentley WaterGEMS V8i updates.
Bentley Institute Training	Opens your browser to the Bentley Institute Training web site.
Bentley Professional Services	Opens your browser to the Bentley Professional Services web site.
Bentley SELECT Support	Opens your browser to SELECTservices area of the Bentley web site.
Bentley Communities	Opens your browser to the BentleyCommunities section of the website.
Bentley.com	Opens the home page on the Bentley web site.
About Bentley WaterGEMS V8i	Opens the About Bentley Bentley WaterGEMS V8i dialog box, which displays copyright information about the product, registration information, and the current version number of the release.



Help Contents View



Help Index View



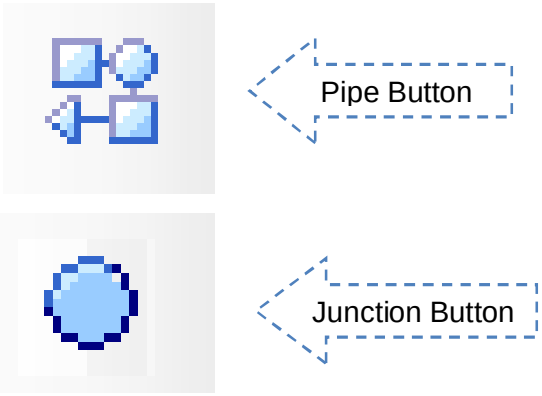
Help Search View

4 BASIC MODEL DEVELOPMENT

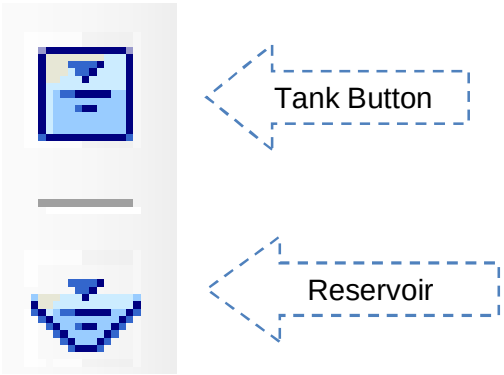
4.1 Building a Network

4.1.1 Add Nodes and Pipes

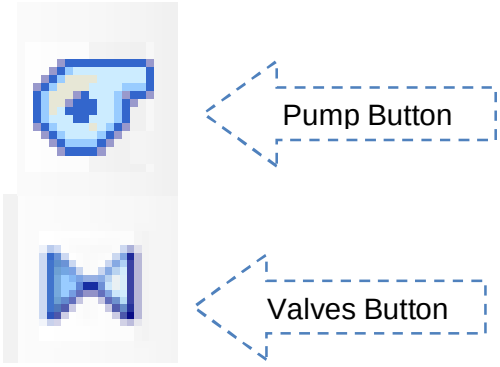
- Add First Node with Junction Button.
- Add subsequent pipes and nodes with network button.



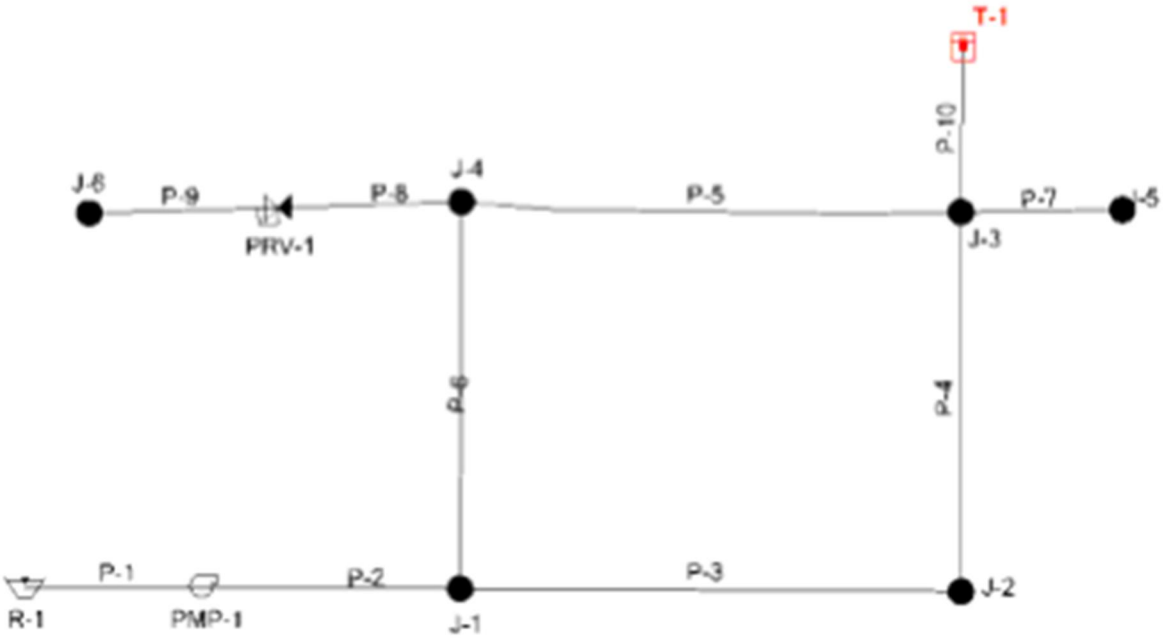
- Right Click and select “Done” when finished.
- Add Tanks and Reservoirs.



- Add Pumps and Valves.



- Complete Network Framework.



The various techniques for adding pipes:

1. Directly between two junctions – double click
2. Click on first junction, click in space to add new junction
3. Click on first junction; right click to select “Bend” (or any other type of element). Switch back to Junction to complete segment

Right click and select “Done”

4.2 Enter and Modify Data

You can use the Select tool and double-click an element to bring up its Properties editor. (Note that if the properties window is already open, a single click with bring up the properties of the element that was clicked on).

Or you can click FlexTables to bring up dynamic tables that allow you to edit and display the model data in a tabular format. You can edit the data as you would in a spreadsheet. (Only cells that do not have yellow highlighting can be edited.)

Using FlexTables you can view input data and results for all elements of a specific type in a tabular format. You can use the standard set of FlexTables or create customized FlexTables to compare data and create reports.

You can view all elements in the project, all elements of a specific type, or any subset of elements. Additionally, to ease data input and present output data for specific elements, FlexTables can be:

- Filtered
- Globally edited
- Sorted
- Sorted.

If you need to edit a set of properties for all elements of a certain type in your network, you can create a FlexTable and make your changes there rather than editing each element one at a time in sequence.

FlexTables can also be used to create results reports that you can print, save as a file, or copy to the Windows clipboard for copying into word processing or spreadsheet software.

To work with FlexTables, select the FlexTables manager or go to **View > FlexTables**<Ctrl+7> to open the FlexTables manager if it is closed.

You can sort and filter your FlexTables to focus on specific data or present your data in one of the following ways:

You can edit a FlexTable to change the columns of data it contains or the values in some of those columns.

Editable columns

Columns that contain data you can edit are displayed with a white background. You can change these columns directly in the FlexTable and your changes are applied to your model when you click **OK**.

The content in the FlexTable columns can be changed in other areas, such as in a Property Editor or managers.

If you make a change that affects a FlexTable outside the FlexTable, the FlexTable is updated automatically to reflect the change.

Non-editable columns

Columns that contain data you cannot edit are displayed with a yellow background and correspond to model results calculated by the program and composite values.

The content in these columns can be changed in other areas, for example a Property Editor or by running a computation.

If you make a change that affects a FlexTable outside the FlexTable, the FlexTable is updated automatically to reflect the change.

To edit a FlexTable

1. Click **View > FlexTables** to open the FlexTables manager, then you can:
    - Right-click the FlexTable, then select **Edit**.
    - Double-click the FlexTable to open it, then click **Edit**.
    - Click the FlexTable to select it, then click the **Edit** button.
2. The Table dialog box opens. .
3. Use the Table dialog box to include and exclude columns and change the order in which the columns appear in the table.
4. Click **OK** after you finish making changes to save your changes and close the dialog box; or click **Cancel** to close the dialog box without making changes.

Editing Column-Heading Text

To change the text of a column heading:

1. Click **View > FlexTables** to open the FlexTables manager.
2. In the FlexTables manager, open the FlexTable you want to edit.
3. Right-click the column heading and select **Edit Column Label**.
4. Type the new name for the label and click **OK** to save those changes and close the dialog box or **Cancel** to exit without making any changes.

Changing Units, Format, and Precision in FlexTables

To change the units, format, or precision in a column of a FlexTable:

1. Click **View > FlexTables** to open the FlexTables manager.
2. In the FlexTables manager, open the FlexTable you want to edit.
3. Right-click the column heading and select **Units**.
4. Make the changes you want and click **OK** to save those changes or **Cancel** to exit without making any changes.

To sort the order of columns in a FlexTable

You can sort the order of columns in a FlexTable in two ways:

- Edit the FlexTable; open the Table dialog box and change the order of the selected tables using the up and down arrow buttons.

  The top-most item in the Selected Columns pane appears furthest to the left in the resulting FlexTable.

- Open the FlexTable, click the heading of the column you want to move, then click again and drag the column to the new position. You can only move one column at a time.

To sort the contents of a FlexTable

1. Open the FlexTable to be edited.
2. Right-click a column heading to rank the contents of the column.
3. Select **Sort** then choose.
    - **Sort Ascending**—Sorts alphabetically from A to Z, from top to bottom. Sorts numerically from negative to positive, from top to bottom. Sorts selected check boxes to the top and cleared ones to the bottom.
    - **Sort Descending**—Sorts alphabetically from Z to A, from top to bottom. Sorts numerically from positive to negative, from top to bottom. Sorts cleared check boxes to the top and selected ones to the bottom.
    - **Custom**—Select one or more sort keys
    - **Reset**—Back to the original sorting order

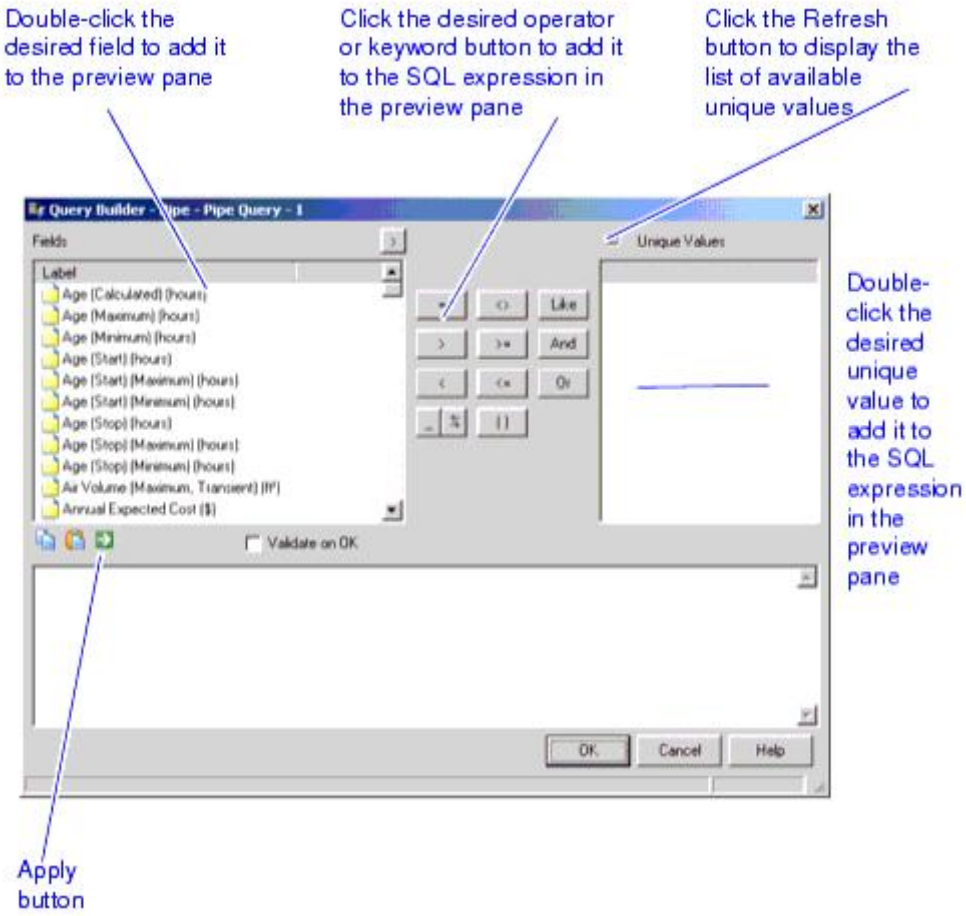
To filter a FlexTable

Filter a FlexTable by creating a query.

1. Open the FlexTable to be filtered.
2. Right-click the column heading to filter and select **Filter**.

   Select **Custom** to open the Query Builder dialog box.

3. All input and results fields for the selected element type appear in the Fields list pane, available SQL operators and keywords are represented by buttons, and available values for the selected field are listed in the Unique Values list pane. Perform the following steps to construct your query:
   a. Double-click the field to include in your query. The database column name of the selected field appears in the preview pane.
   b. Click the desired operator or keyword button. The SQL operator or keyword is added to the SQL expression in the preview pane.
   c. Click the **Refresh** button above the Unique Values list pane to see a list of unique values available for the selected field. The **Refresh** button becomes disabled after you use it for a particular field.
   d. Double-click the unique value you want to add to the query. The value is added to the SQL expression in the preview pane.
   e. Click **Apply**  above the preview pane to validate your SQL expression. If the expression is valid, the window "Query Successful" opens. Click **OK**. The word **VALIDATED** will be at the bottom of the window.



f. Click **OK**.

The FlexTable displays columns of data for all elements returned by the query and the word "FILTERED" is displayed in the FlexTable status bar.

The status pane at the bottom of the Table window always shows the number of rows displayed and the total number of rows available (for example, 10 of 20 elements displayed).

If you change the values for an attribute that is being sorted or filtered, the sort or filter operation needs to be reapplied. To do this, use the Apply Sort/Filter command accessible from the right-click context menu.

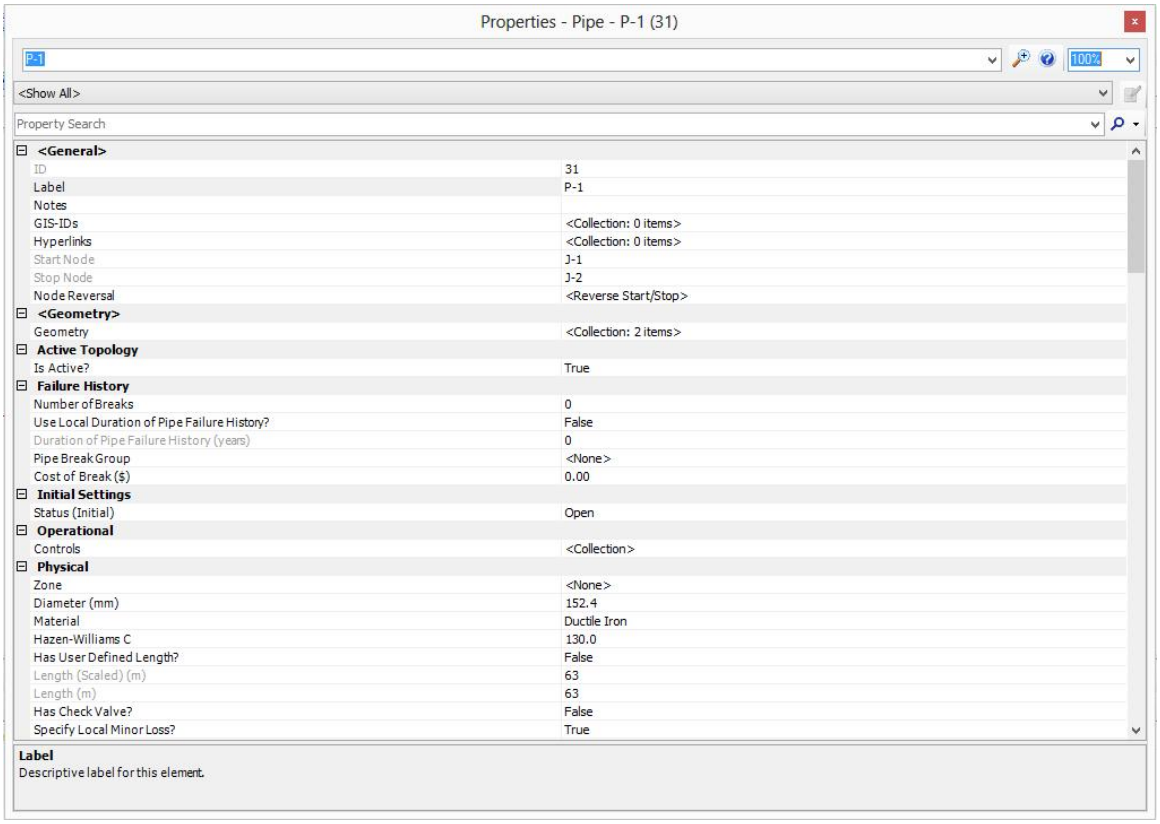
To reset a filter

1. Right-click the column heading you want to filter.
2. Select **Filter**.
3. Click Reset.
4. Click **Yes** to reset the active filter.

To reapply a sort or filter operation

1. Right-click the column heading for the sort or filter operation you want reapplied.
2. Select Apply Sort/Filter.

4.3 Pipe Data Input

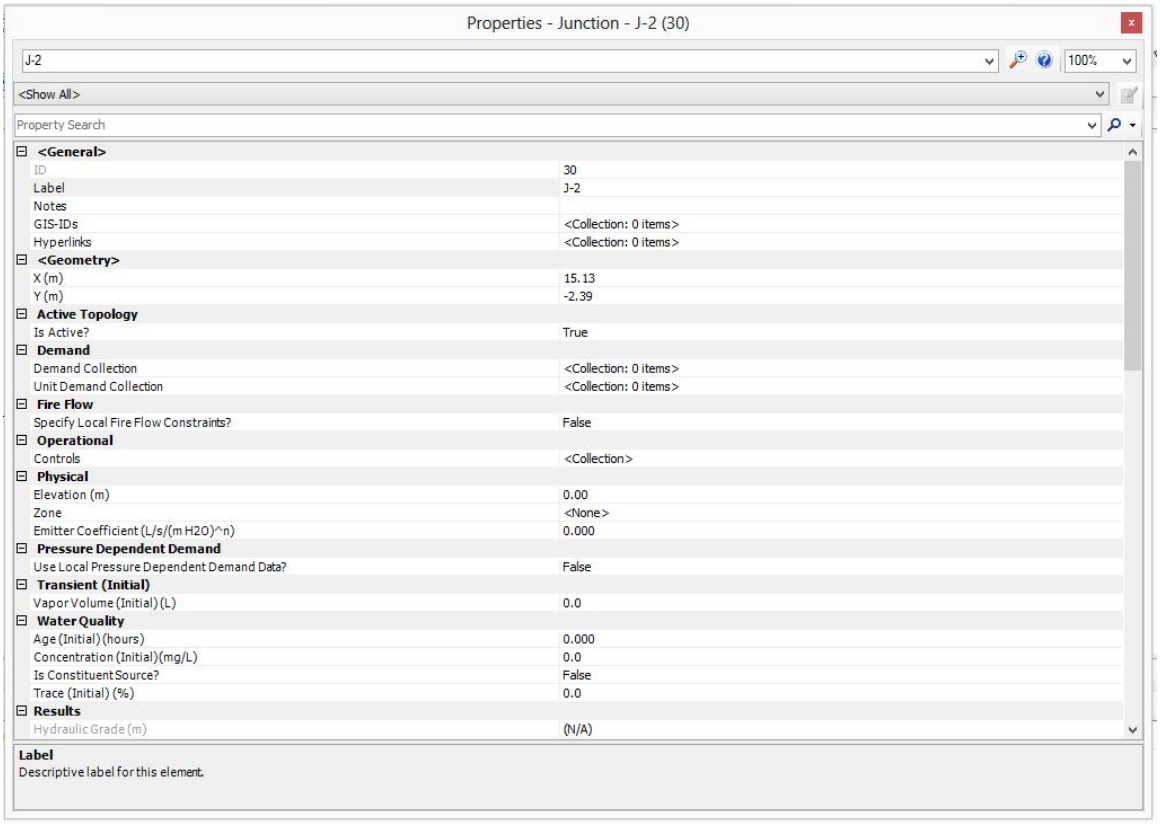


- Double Click on Pipe.
- Or Click “CTRL-F” and type in Pipe ID to find pipe.

The key pipe characteristics to be entered:

- Diameter (mandatory)
- Roughness (mandatory)
- Minor Losses
- Zone
- Material
- User-defined length (if needed)
- Installation Year.

4.4 Junction Data Input

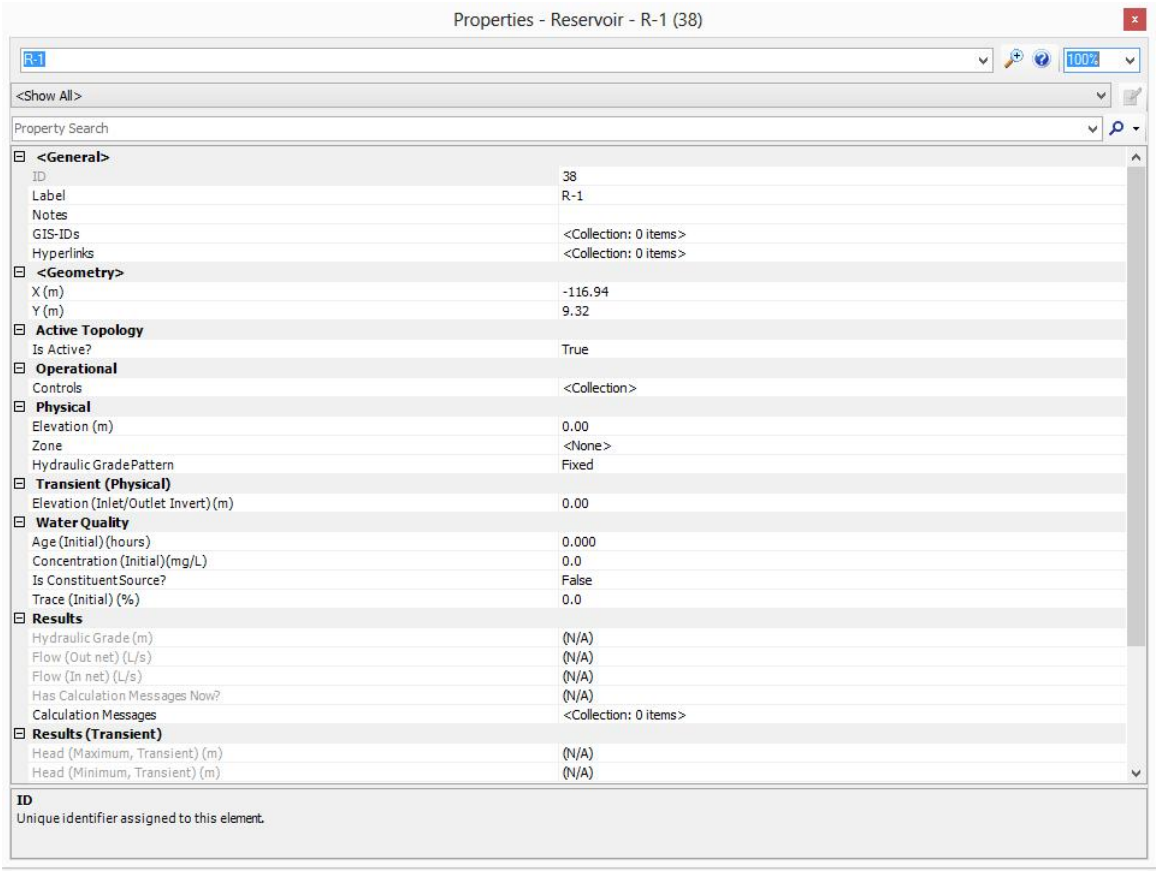


- General Data.
- Add Elevation.
- Designate pressure zone.
- Assign demand.

The key Junction characteristics to be entered:

- Elevation (mandatory)
- Demands (mandatory)
- Zone

4.5 Reservoir Data Input

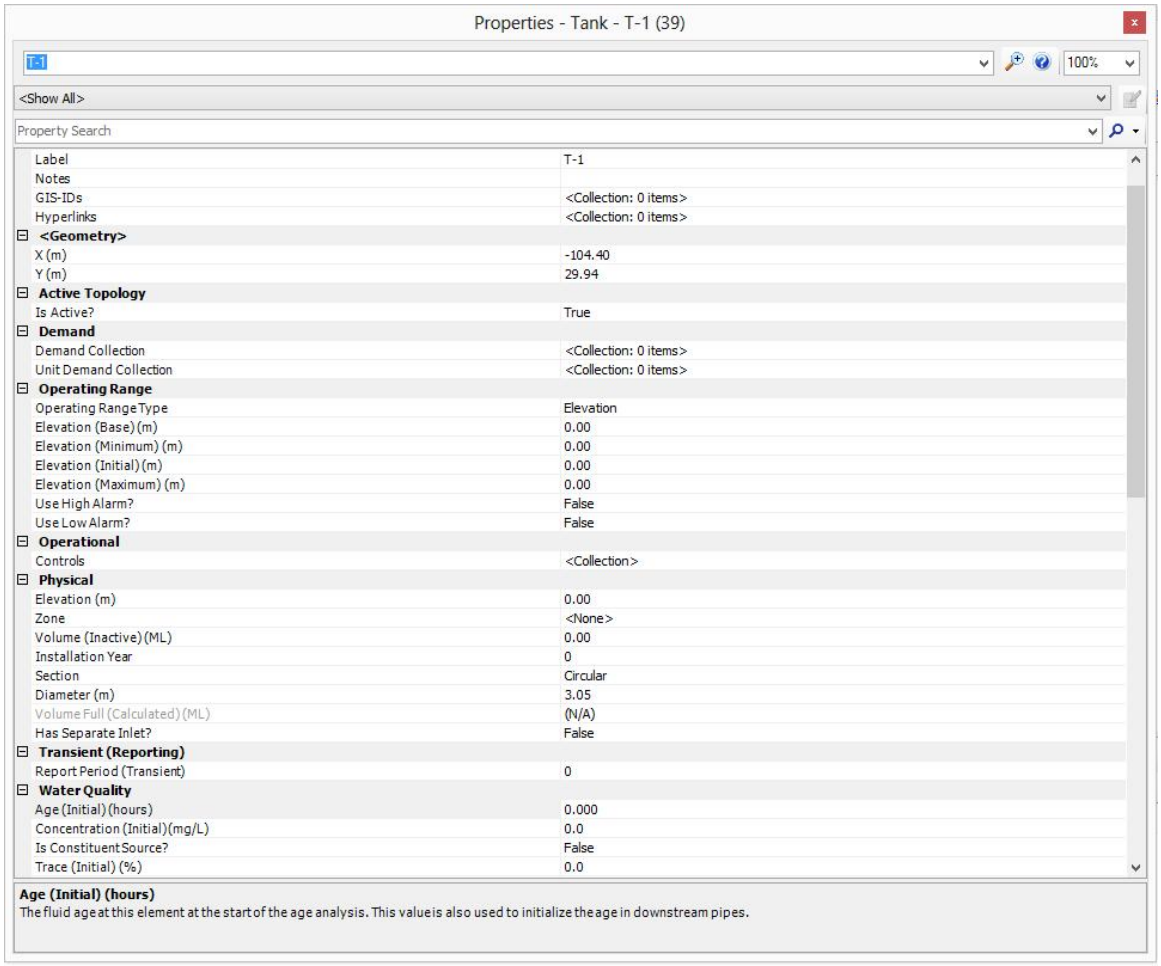


- Edit Reservoir Data.

The key Reservoir characteristics to be entered:-

- Elevation
- Zone
- Inlet elevation
- HGL Pattern

4.6 Tank Data Input

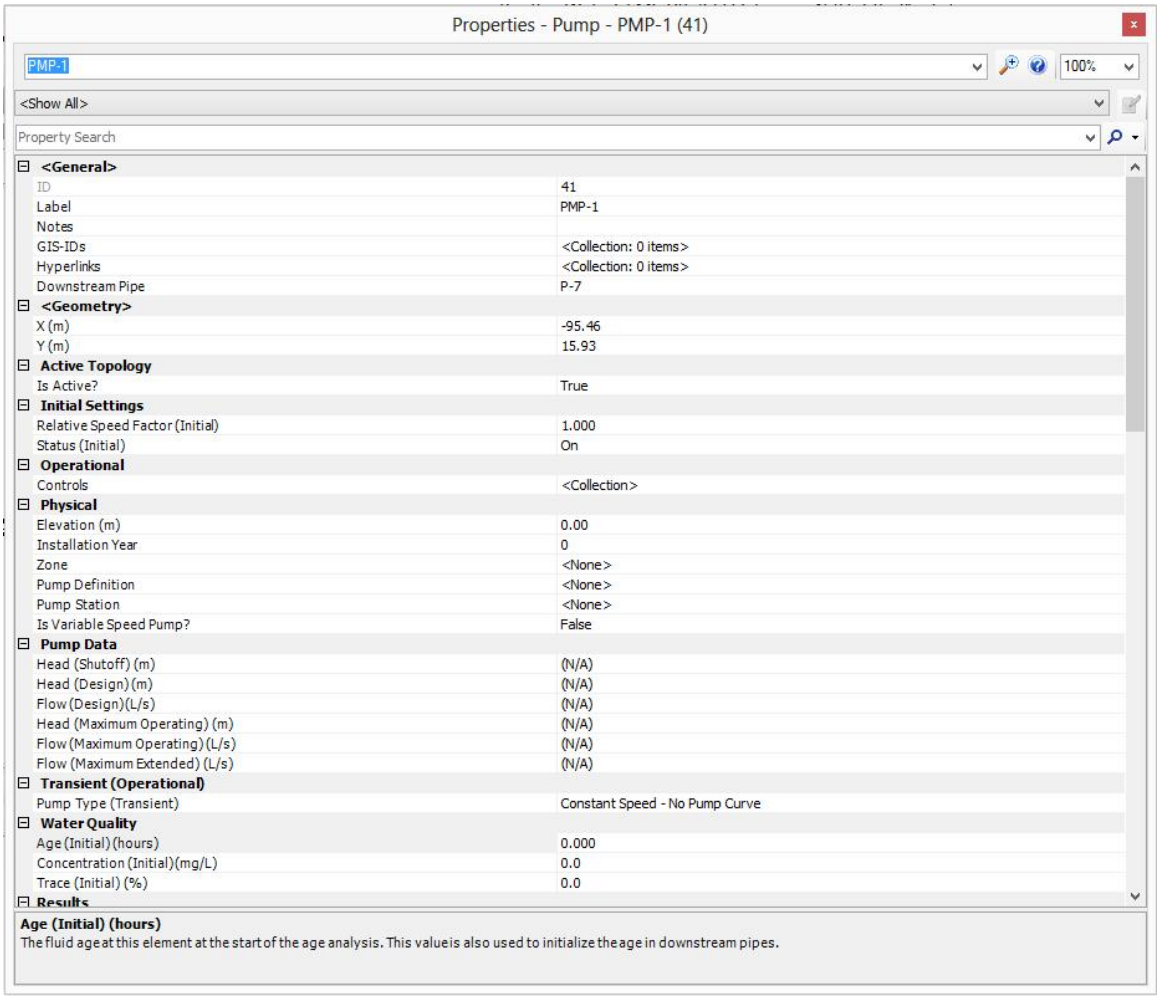


- Edit Tank Data.

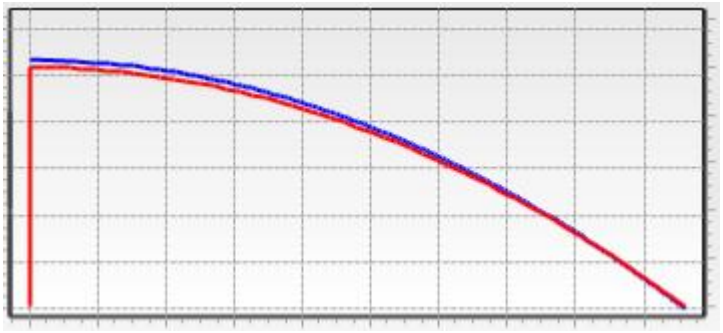
The key Tank characteristics to be entered:

- Elevations or levels (base, initial, maximum)
- Volume
- Shape
- Installation Year

4.7 Pump Data Input



- Set Pump Elevation
- Define Pump definition by clicking on Pump Definition



Pump Definition curve.

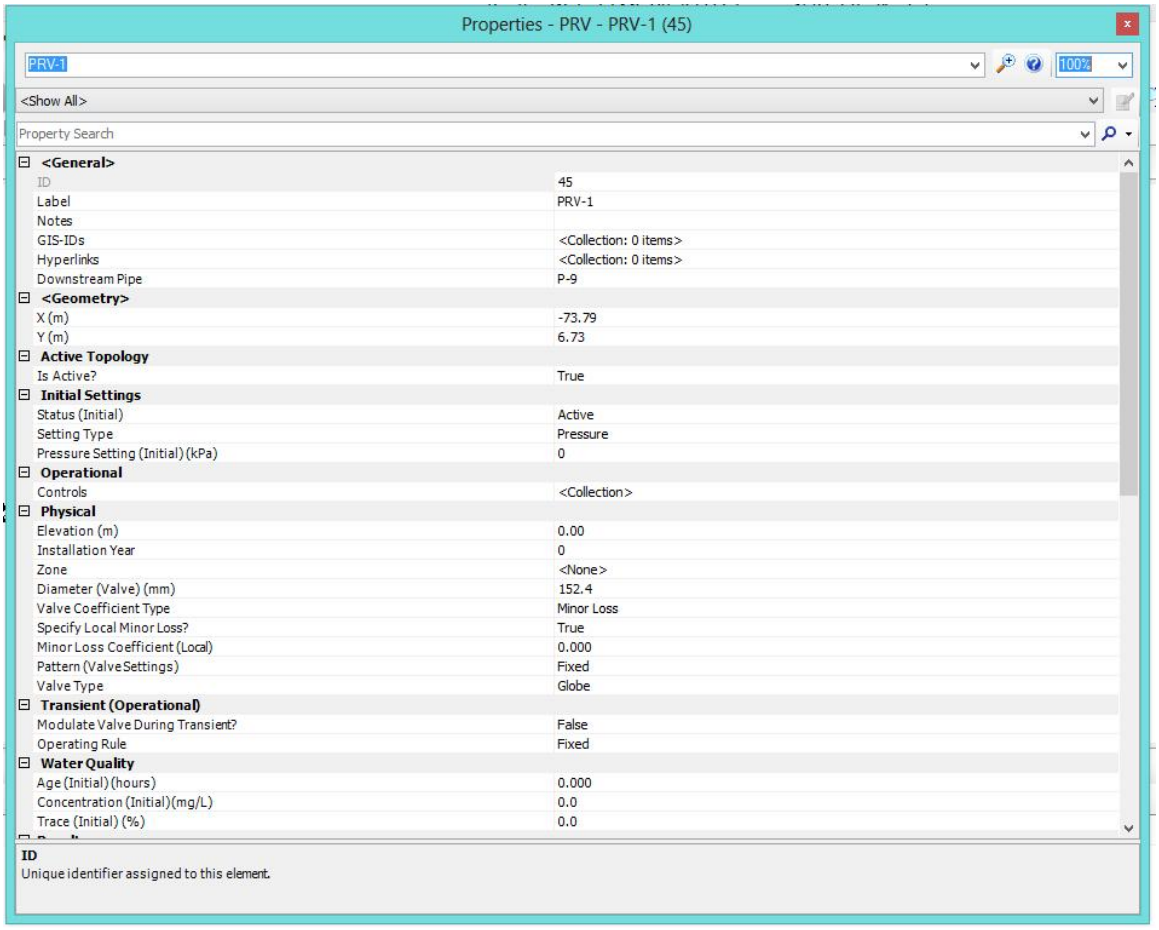
The key Pump characteristics to be entered:

- Speed Factor
- Status
- Elevation

- Installation Year
- Zone
- Pump Definition (More detail on pump definition options)
- Is Variable Speed?

4.8 Valve Data Input

- Select Type of Valve from Valve button.
- Types:
  - Pressure Reducing
  - Pressure Sustaining
  - Pressure Breaking
  - Flow Control
  - Throttle Control
  - General Purpose
- Minor Loss Coefficient is for full open valve
- Settings vary by type of valve – PRV's set by pressure



- Minor Loss coefficient
- Setting
- Setting Pattern

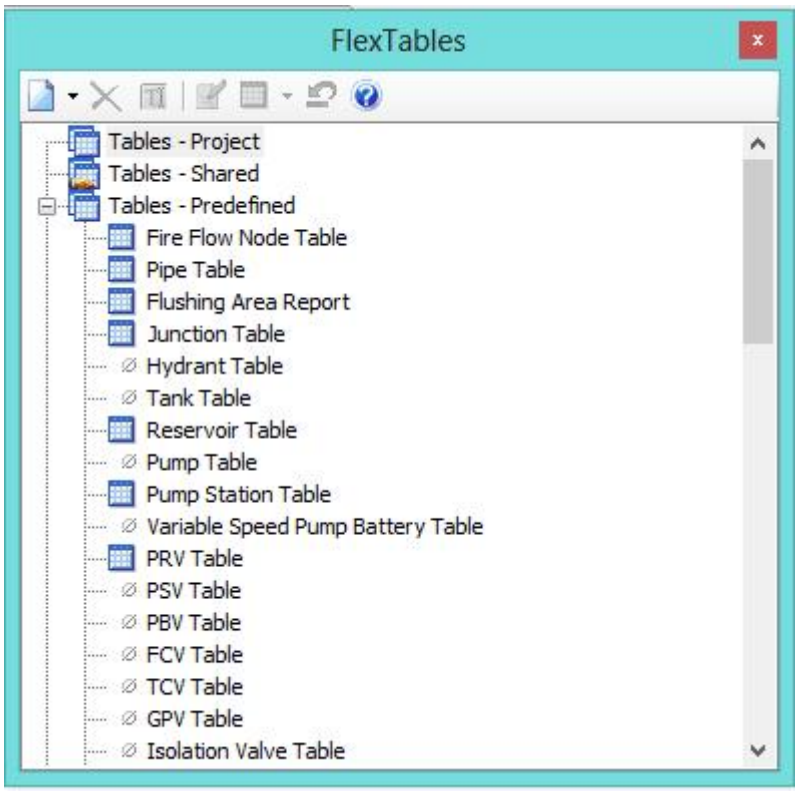
Note: The different types of valve are described in the following table:

Valve Type	Description
Pressure Reducing Valve (PRV)	PRVs throttle to prevent the downstream hydraulic grade from exceeding a set value. If the downstream grade rises above the set value, the PRV will close. If the head upstream is lower than the valve setting, the valve will open fully.
Pressure Sustaining Valve (PSV)	A Pressure Sustaining Valve (PSV) is used to maintain a set pressure at a specific point in the pipe network. The valve can be in one of three states: • partially opened (i.e., active) to maintain its pressure setting on its upstream side when the downstream pressure is below this value • fully open if the downstream pressure is above the setting • closed if the pressure on the downstream side exceeds that on the upstream side (i.e., reverse flow is not allowed).
Pressure Breaker Valve (PBV)	PBVs are used to force a specified pressure (head) drop across the valve. These valves do not automatically check flow and will actually boost the pressure in the direction of reverse flow to achieve a downstream grade that is lower than the upstream grade by a set amount.
Flow Control Valve (FCV)	FCVs are used to limit the maximum flow rate through the valve from upstream to downstream. FCVs do not limit the minimum flow rate or negative flow rate (flow from the To Pipe to the From Pipe).
Throttle Control Valve (TCV)	TCVs are used as controlled minor losses. A TCV is a valve that has a minor loss associated with it where the minor loss can change in magnitude according to the controls that are implemented for the valve. If you don't know the headloss coefficient, you can also use the discharge coefficient, which will be automatically converted to an equivalent headloss coefficient in the program. To specify a discharge coefficient, change the Coefficient Type to Discharge Coefficient.
General Purpose Valve (GPV)	GPVs are used to model situations and devices where the flow-to-headloss relationship is specified by you rather than using the standard hydraulic formulas. GPVs can be used to represent reduced pressure backflow prevention (RPBP) valves, well draw-down behavior, and turbines.

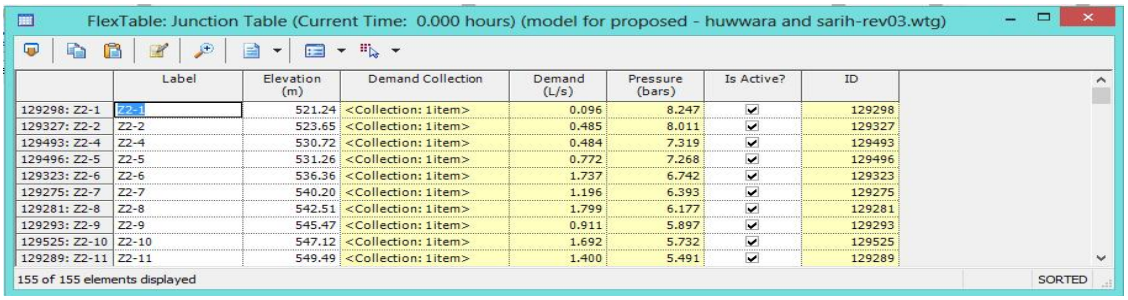
Note: Once the type of valve is chosen, the valve type options are as follows: butterfly, needle, circular gate, globe, ball, and user-defined.

4.9 Editing in Tables

You can click FlexTables  to bring up dynamic tables that allow you to edit and display the model data in a tabular format. You can edit the data as you would in a spreadsheet.



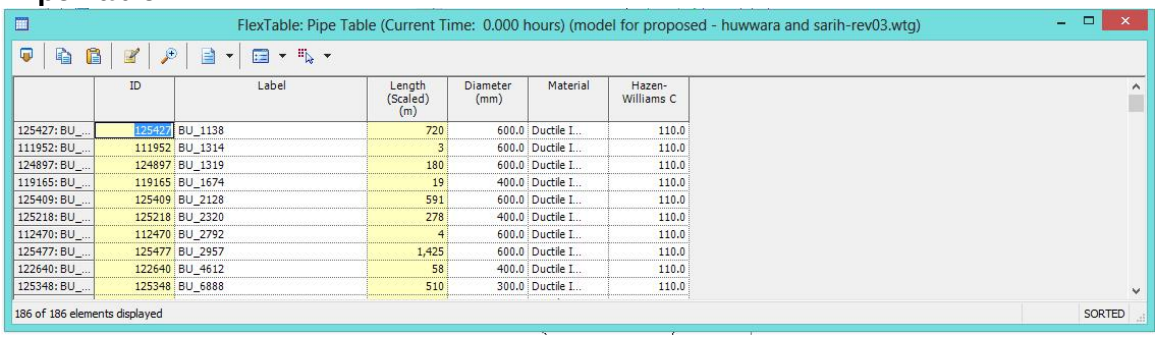
Junction Table


FlexTable: Junction Table (Current Time: 0.000 hours) (model for proposed - huwwara and sarih-rev03.wtg)

	Label	Elevation (m)	Demand Collection	Demand (L/s)	Pressure (bars)	Is Active?	ID
129298: Z2-1	Z2-1	521.24	<Collection: 1 item>	0.096	8.247	☑	129298
129327: Z2-2	Z2-2	523.65	<Collection: 1 item>	0.485	8.011	☑	129327
129493: Z2-4	Z2-4	530.72	<Collection: 1 item>	0.484	7.319	☑	129493
129496: Z2-5	Z2-5	531.26	<Collection: 1 item>	0.772	7.268	☑	129496
129323: Z2-6	Z2-6	536.36	<Collection: 1 item>	1.737	6.742	☑	129323
129275: Z2-7	Z2-7	540.20	<Collection: 1 item>	1.196	6.393	☑	129275
129281: Z2-8	Z2-8	542.51	<Collection: 1 item>	1.799	6.177	☑	129281
129293: Z2-9	Z2-9	545.47	<Collection: 1 item>	0.911	5.897	☑	129293
129525: Z2-10	Z2-10	547.12	<Collection: 1 item>	1.692	5.732	☑	129525
129289: Z2-11	Z2-11	549.49	<Collection: 1 item>	1.400	5.491	☑	129289

155 of 155 elements displayed SORTED

Pipe Table


FlexTable: Pipe Table (Current Time: 0.000 hours) (model for proposed - huwwara and sarih-rev03.wtg)

	ID	Label	Length (Scaled) (m)	Diameter (mm)	Material	Hazen-Williams C
125427: BU_...	125427	BU_1138	720	600.0	Ductile I...	110.0
111952: BU_...	111952	BU_1314	3	600.0	Ductile I...	110.0
124897: BU_...	124897	BU_1319	180	600.0	Ductile I...	110.0
119165: BU_...	119165	BU_1674	19	400.0	Ductile I...	110.0
125409: BU_...	125409	BU_2128	591	600.0	Ductile I...	110.0
125218: BU_...	125218	BU_2320	278	400.0	Ductile I...	110.0
112470: BU_...	112470	BU_2792	4	600.0	Ductile I...	110.0
125477: BU_...	125477	BU_2957	1,425	600.0	Ductile I...	110.0
122640: BU_...	122640	BU_4612	58	400.0	Ductile I...	110.0
125348: BU_...	125348	BU_6888	510	300.0	Ductile I...	110.0

186 of 186 elements displayed SORTED

Editable columns

Columns that contain data you can edit are displayed with a white background. You can change these columns directly in the FlexTable and your changes are applied to your model when you click OK.

The content in the FlexTable columns can be changed in other areas, such as in a Property Editor or managers.

If you make a change that affects a FlexTable outside the FlexTable, the FlexTable is updated automatically to reflect the change.

Non-editable columns

Columns that contain data you cannot edit are displayed with a yellow background and correspond to model results calculated by the program and composite values.

The content in these columns can be changed in other areas, for example a Property Editor or by running a computation.

If you make a change that affects a FlexTable outside the FlexTable, the FlexTable is updated automatically to reflect the change.

To edit a FlexTable

1. Click View > FlexTables to open the FlexTables manager, then you can:
   - Right-click the FlexTable, then select Edit.
   - Double-click the FlexTable to open it, then click Edit.
   - Click the FlexTable to select it, and then click the Edit button.
2. The Table dialog box opens. .
3. Use the Table dialog box to include and exclude columns and change the order in which the columns appear in the table.

4. Click OK after you finish making changes to save your changes and close the dialog box; or click Cancel to close the dialog box without making changes.

Editing Column-Heading Text
To change the text of a column heading:

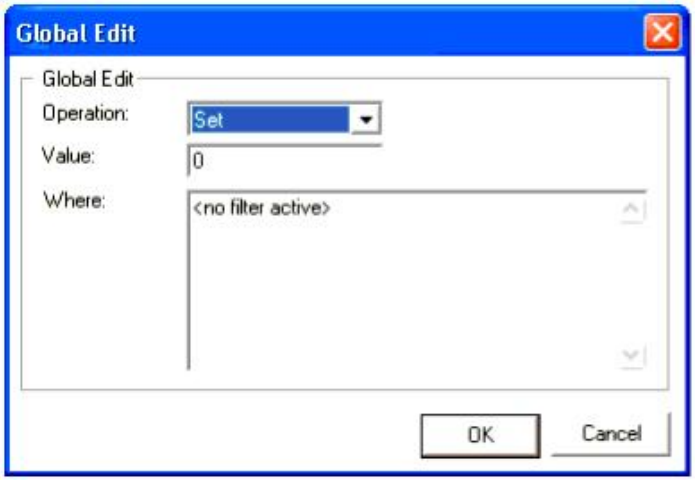
1. Click View > FlexTables to open the FlexTables manager.
2. In the FlexTables manager, open the FlexTable you want to edit.
3. Right-click the column heading and select Edit Column Label.
4. Type the new name for the label and click OK to save those changes and close the dialog box or Cancel to exit without making any changes.

Changing Units, Format, and Precision in FlexTables to change the units, format, or precision in a column of a FlexTable

1. Click View > FlexTables to open the FlexTables manager.
2. In the FlexTables manager, open the FlexTable you want to edit.
3. Right-click the column heading and select Units.
4. Make the changes you want and click OK to save those changes or Cancel to exit without making any changes.

Navigating in Tables
The arrow keys, <Ctrl+End>, <Page Up>, <Page Down>, and <Ctrl+arrow> keys navigate to different cells in a table.

Globally Editing Data
Using FlexTables, you can globally edit all of the values in an entire editable column. Globally editing a FlexTable column can be more efficient for editing properties of an element than using the Properties Editor or managers to edit each element in your model individually.



Operation	Select the type of edit to perform: • Set: Changes each of the entries in the column to the value in the Value box. • Add: Adds the value in the Value box to each of the entries in the column. • Divide: Divides each of the entries in the column by the value in the Value box. • Multiply: Multiplies each of the entries in the column by the value in the Value box. • Subtract: Subtracts the value in the Value box from each of the entries in the column.
Value	Type the value that will be used in the chosen Operation to edit the entries of the column.
Where	When the Table has an active filter, the SQL Query used by the filter is displayed in this pane.

To globally edit the values in a FlexTable column

1. Click View > FlexTables to open the FlexTables manager.
2. In the FlexTables manager, open the FlexTable you want to edit and find the column of data you want to change.

   If necessary, you might need to first create a FlexTable or edit an existing one to make sure it contains the column you want to change.

3. Right-click the column heading and select Global Edit.
4. In the Operation field, select what you want to do to data in the column: Add, Divide, Multiply, Set, or Subtract.

   Note: The Operation field is only available for numeric data.

5. In the Global Edit field, type or select the value.

5 MODELBUILDER APPLICATION

5.1 Using ModelBuilder to Transfer Existing Data

ModelBuilder lets you use your existing GIS asset to construct a new WaterGEMS V8i model or update an existing WaterGEMS V8i model. ModelBuilder supports a wide variety of data formats, from simple databases (such as Access and DBase), spreadsheets (such as Excel), and GIS data (such as shape files), to high end data stores (such as Oracle, and SQL Server), and more.

Using ModelBuilder, you map the tables and fields contained within your data source to element types and attributes in your WaterGEMS V8i model. The result is that a WaterGEMS V8i model is created. ModelBuilder can be used in any of the Bentley WaterGEMS V8i platforms - Stand-Alone, MicroStation mode, AutoCAD mode, or ArcGIS mode.

ModelBuilder is the first tool you will use when constructing a model from GIS data. The steps that you take at the outset will impact how the rest of the process goes. Take the time now to ensure that this process goes as smoothly and efficiently as possible. (Note: retain as much data as possible from the native data in the model data to facilitate sharing of data between sources in the future.)

The GIS-ID Property
All elements in WaterGEMS V8*i* have an editable GIS-ID property which can be used for maintaining associations between records in your source file and elements in your model. These associations can be one-to-one, one-to-many, or many-to-one.

ModelBuilder can take advantage of this GIS-ID property, and has advanced logic for keeping your model and GIS source file synchronized across the various model to GIS associations.

The GIS-ID is a unique field in the source file which the user selects when ModelBuilder is being set up. In contrast to using Label (which is adequate if model building is a one-time operation) as the key field between the model and the source file, a GIS-ID has some special properties which are very helpful in maintaining long-term updating of the model as the data source evolves over time.

In addition, WaterGEMS V8*i* will intelligently maintain GIS-ID as you use the various tools to manipulate elements (Delete, Morph, Split, Merge Nodes in Close Proximity).

- When an element with one or more GIS-IDs is deleted, ModelBuilder will not recreate it the next time a synchronization from your GIS occurs if the "Recreate elements associated with a GIS-ID that was previously deleted from the model" option is left unchecked.
- When an element with one or more GIS-IDs is morphed, the new element will preserve those GIS-IDs. The original element will be considered as "deleted with GIS-IDs", which means that it will not be recreated by default (see above).
- When a link is split, the two links will preserve the same GIS-IDs that the original pipe had. On subsequent ModelBuilder synchronizations, any data-change occurring for the associated record in the GIS can be cascaded into all the split link segments (see ModelBuilder - additional options).

- When nodes in close proximity are merged, the resulting node will preserve the GIS-IDs of all the nodes that were removed. On subsequent ModelBuilder synchronizations into the model, if there are data-update conflicts between the records in the GIS associated with the merged node in the model, updates from the first GIS-ID listed for the merged node will be preserved in the model. Note that in this case, the geometry of the merged node cannot be updated in the model. For synchronizations going from the model to the GIS, data-updates affecting merged-nodes can be cascaded into all the associated records in the GIS (see ModelBuilder - additional options).

To support these relationship (specifically one to many), GIS-ID are managed as a collection property (capable of holding any number of GIS identifiers).

A variety of model element(s) to GIS record(s) associations can be specified:

- If the GIS-ID collection is empty, there is no association between the GIS and this element.
- If there is a single entry, this element is associated with one record in the GIS.
- If there are multiple entries, this element is associated with multiple records in the GIS.
- More than one element in the model can have the same GIS-ID, meaning multiple records on the model are associated with a single record in the GIS.

Note: You can also manually edit the GIS-ID property to review or modify the element to GIS associations

5.2 Preparing to Use ModelBuilder

- **Determine the purpose of your model** – Once you establish the purpose of your model, you can start to make decisions about how detailed the model should be.

- **Get familiar with your data –** ModelBuilder supports several data source types, including tabular and geometric. Tabular data sources include spreadsheets, databases, and other data sources without geometric information. Some supported tabular data source types include Microsoft Excel, and Microsoft Access files. Geometric data sources, while also internally organized by tables, include geometric characteristics such as shape type, size, and location. Some supported geometric data source types include the major CAD and GIS file types.

  If you obtained your model data from an outside source, take the time to get acquainted with it in its native platform. For example, review spatial and attribute data directly in your GIS environment. Do the nodes have coordinate information, and do the pipes have start and stop nodes specified? If not, the best method of specifying network connectivity must be determined.

  Contact those involved in the development of the GIS to learn more about the GIS tables and associated attributes. Find out the purpose of any fields that may be of interest, ensure that data is of an acceptable accuracy, and determine units associated with fields containing numeric data.

  Ideally, there will be one source data table for each WaterGEMS V8i element type. However, this is not always the case, and there are two other possible scenarios:

Many tables for one element type – In this case, there may be several tables in the datasource corresponding to a single GEMS modeling element, component, or collection. In this case each data source table must be individually mapped to the WaterGEMS V8i table type, or the tables must be combined into a single table from within its native platform before running ModelBuilder.

One table containing many element type – In this case, there may be entries that correspond to several WaterGEMS V8i table types in one datasource table. You should separate these into individual tables before running ModelBuilder. The one case where a single table can work is when the features in the table are ArcGIS subtypes. ModelBuilder handles these subtypes by treating them as separate tables when setting up mappings. Shapefile are an acceptable ArcGIS data source (Note that Shapefiles can be converted into Geodatabase Feature Classes if you would like to make use of Subtypes.)

If multiple types of WaterGEMS V8*i* elements have their data stored in a single geodatabase table, then each element must be a separate ArcGIS subtype. For example, in a valve table PRVs may be subtype 1, PSVs may be subtype 2, FCVs may be subtype 3, and so on. With subtypes, it is not necessary to follow the rule that each GIS/database feature type must be associated with a single type of GEMS model element. Note that the subtype field must be of the integer type (e.g., 1, 2) and not an alphanumeric field (e.g., PRV). For more information about subtypes, see ArcGIS Help.

ModelBuilder has built in support for subtypes. After selecting your data source, feature classes will automatically be categorized by subtype. This gives you the ability to assign mappings at the subtype level. For example, ModelBuilder allows you to exclude a particular subtype within a feature class, or associate each subtype with a different element type.

Note: If you are working with an ArcGIS data source, note that ModelBuilder can only use geodatabases, geometric networks, and coverages in ArcGIS mode.

Note: Source data must be "clean" e.g. no blank columns or rows, or empty key fields.

Note: It is important to insure that any shapefile field names are less than or equal to 10 characters.

Note: In some versions of GIS, there are restrictions on use of characters or on the length of the field name.

- **All mappings should be contained in a single ModelBuilder connection** – ModelBuilder will ensure that data is synchronized into the model in the correct order using this technique. If multiple connections are to be used instead, then the user should run the individual ModelBuilder connections to get the following data synchronization order: Components, Nodes, Pipes, polygon data (if any), Directed Nodes (i.e. node types with a Downstream Pipe field), and finally collection data.

Note: Pipes are brought in first if no node information is available.)

- **Preparing your data** – When using ModelBuilder to get data from your data source into your model, you will be associating rows in your data source to elements in WaterGEMS V8i. Your data source needs to contain a Key/Label field that can be

used to uniquely identify every element in your model. The data source tables should have identifying column labels, or ModelBuilder will interpret the first row of data in the table as the column labels. Be sure data is in a format suited for use in ModelBuilder. Where applicable, use powerful GIS and Database tools to perform Database Joins, Spatial Joins, and Update Joins to get data into the appropriate table, and in the desired format.

Note: When working with ID fields, the expected model input is the WaterGEMS V8i ID. After creating these items in your WaterGEMS V8i model, you can obtain the assigned ID values directly from your WaterGEMS V8i modeling file. Before synchronizing your model, get these WaterGEMS V8i IDs into your data source table (e.g., by performing a database join).

- **Preparing your CAD Data**– In previous versions of WaterGEMS V8i, the Polyline-to-Pipe feature was used to import CAD data into a WaterGEMS V8i model. In v8, CAD data is imported using ModelBuilder. When using ModelBuilder to import data from your CAD file into your model, you will be associating layers in your CAD drawing with elements in WaterGEMS V8i.

  Different CAD layers will be recognized as different element types and presented as tables existing in your CAD data source. It is recommended that you natively export your AutoCAD .dwg or MicroStation .dgn files first as a .dxf file, then select this .dxf as the data source in ModelBuilder. Your data source will most likely not contain a Key/Label field that can be used to uniquely identify every element in your model, so ModelBuilder will automatically generate one for you using the default "<label>". This "<label>" field is a combination of an element's cell type label, its shape type, and a numeric ID that represents the order in which it was created. (Note: setting defaults or prototypes for the pipes in WaterGEMS can help with importing CAD data.)

- **Build first, Synchronize later** – ModelBuilder allows you to construct a new model or synchronize to an existing model. This gives you the ability to develop your model in multiple passes. On the first pass, use a simple connection to build your model. Then, on a subsequent pass, use a connection to load additional data into your model, such as supporting pattern or collection data.

Note: Upon completion of your ModelBuilder run, it is suggested you use the Network Navigator to identify any connectivity or topological problems in your new model. For instance, Pipe Split Candidates can be identified and then automatically modified with the Batch Split Pipe Tool (see Batch Pipe Split Dialog Box).

- **Going Beyond ModelBuilder** – Keep in mind that there are additional ways to get data into your model. ModelBuilder can import loads if you have already assigned a demand to each node. If, however, this information is not available from the GIS data, or if your ldemand data is in a format unrecognized by ModelBuilder (meter data, etc.), use LoadBuilder; this module is a specialized tool for getting this data into your model. In addition, with its open database format, WaterGEMS V8i gives you unprecedented access to your modeling data.

  One area of difficulty in building a model from external data sources is the fact that unless the source was created solely to support modeling, it most likely contains much more detailed information than is needed for modeling. This is especially true with regard to the number of piping elements. It is not uncommon for the data sources to include every service line and hydrant lateral. Such information is not needed for most modeling applications and should be removed to improve model run time,

reduce file size, and save costs. So it is advised to filter GIS data first to prevent extraneous pipes from being imported.

- **Importing Collections** – When you are importing a collection, values will always override existing collection items in the model. In order to preserve existing items, they need to be combined with the new values and imported together.

  For example, importing "Junction, Demand Collection", incoming demand rows will override the existing demand collection, not append to it.

If you want to keep the existing demands, you should first export those values (copy-paste is usually easiest) to your data source (e.g. spreadsheet, shapefile) and make those demands part of the data you are importing. In this way ModelBuilder will import both the original and new demands. (Demands are usually allocated using WaterGEMS tools using the Loadbuilder as the built-in method.)

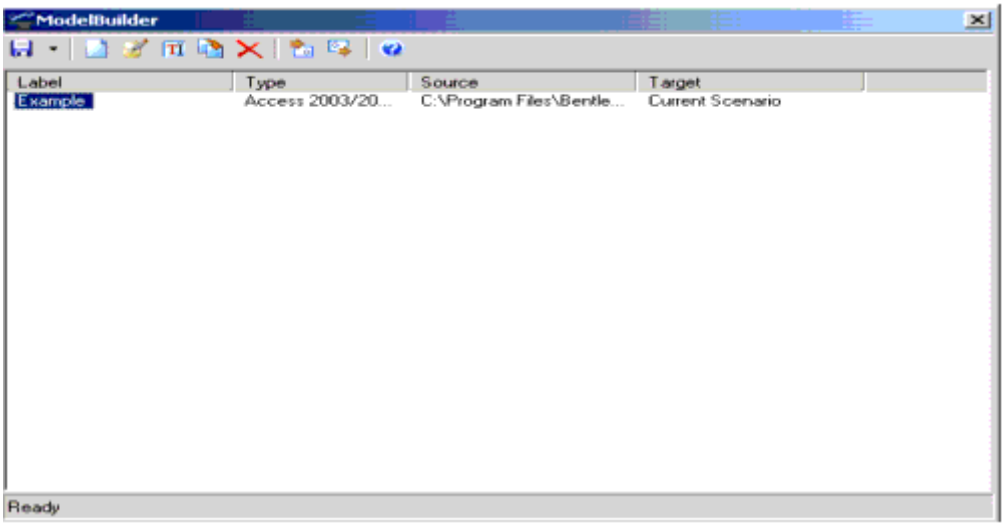
5.3 ModelBuilder Connections Manager

ModelBuilder can be used in any of the Bentley WaterGEMS V8i platforms - Stand-Alone, MicroStation mode, AutoCAD mode, or ArcGIS mode. However, the standalone version of WaterGEMS does not easily support coordinate system but ArcGIS version does.

To access ModelBuilder: Click the Tools menu and select the ModelBuilder command, or click the ModelBuilder button. 

The ModelBuilder Connections manager allows you to create, edit, and manage ModelBuilder connections to be used in the model-building/model-synchronizing process. Each item in this manager represents a "connection" which contains the set of directions for moving data between a source to a target. ModelBuilder connections are not stored in a particular project, but are stored in an external xml file, with the following path:

Windows 7:
C:\Users\<username>\AppData\Roaming\Bentley\<productname>\<productversion>\ModelBuilder.xml



The center of this window is the **Connections List** which displays the list of connections that you have defined. A **toolbar is** located along the top of the Connections list.

The set of buttons on the left of the toolbar allows you to manage your connections:

	**Import/Export**	Click this button to import or export a ModelBuilder Connection file (mbc).
	**New**	Create a new connection using the ModelBuilder Wizard.
	**Edit**	Edit the selected connection using the ModelBuilder Wizard.
	**Rename**	Rename the selected connection.
	**Duplicate**	Create a copy of the selected connection.
	**Delete**	Permanently remove the selected connection.
	**Build Model**	Starts the ModelBuilder build process using the selected connection. This is also referred to as "synching in" from an external data source to a model. Excluding some spatial option overrides, a build operation will update your model with new elements, components, and collections that already exist in the model. Only table types and fields that are mapped will be updated. Depending upon the configuration of synchronization options in the selected connection, if an element in your data source does not already exist in your model, it may be created. If the element exists, only the fields mapped for that table type may be updated. ModelBuilder will not override element properties not specifically associated with the defined field mappings. A Build Model operation will update existing or newly created element values for the current scenario/alternative, or you can optionally create new child scenario/alternatives to capture any data difference.
	**Sync Out**	Starts the ModelBuilder synchronize process using the selected connection. Unless specifically overridden, a Sync Out operation will only work for existing and new elements. On a Sync Out every element in your target data source that also exists in your model will be refreshed with the current model values. If your model contains elements that aren't contained in your data source, those data rows can optionally be added to your target data file. Only those properties specified with field mappings will be synchronized out to the data source. A Sync Out operation will refresh element properties in the data source with the current model values for the current scenario/alternative.
	**Help**	Displays online help.

After initiating a Build or Sync command, ModelBuilder will perform the selected operation. During the process, a progress-bar will be displayed indicating the step that ModelBuilder is currently working on.

When ModelBuilder completes, you will be presented with a summary window that outlines important information about the build process. We recommend that you save this summary so that you can refer to it later.

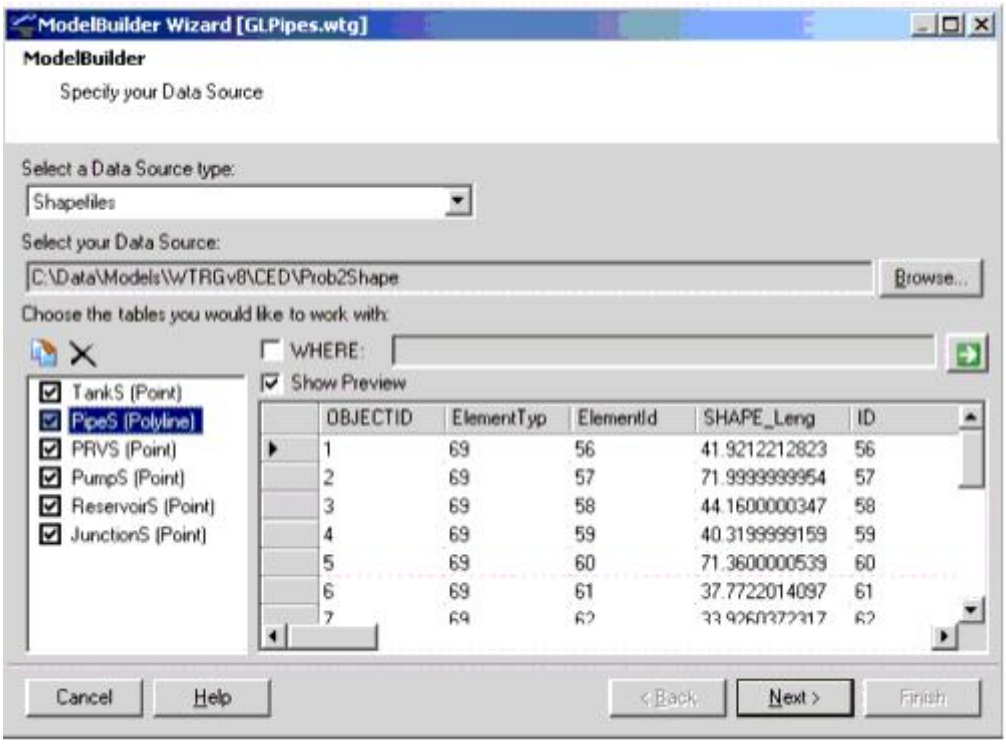
Note: Because the connections are stored in a separate xml file rather than with the project file, ModelBuilder connections are preserved even after Bentley WaterGEMS V8i is closed.

5.4 ModelBuilder Wizard

The ModelBuilder Wizard assists in the creation of ModelBuilder connections. The Wizard will guide you through the process of selecting your data source and mapping that data to the desired input of your model.

5.4.1 Step 1 – Specify Data Source

In this step, the data source type and location are specified. After selecting your data source, the desired database tables can be chosen and previewed.



The following fields are available:

- **Data Source** type (drop-down list) –This field allows you to specify the type of data you would like to work with.

Note: If your specific data source type is not listed in the Data Source type field, try using the OLE DB data source type. OLE DB can be used to access many database systems (including ORACLE and SQL Server).

- **Data Source** (text field) – This read-only field displays the path to your data source.
- **Browse** (button) – This button opens a browse dialog box that allows you to interactively select your data source.

Note: Some Data Source types expect you to choose more than one item in the Browse dialog box. For more information, see Multi-select Data Source Types.

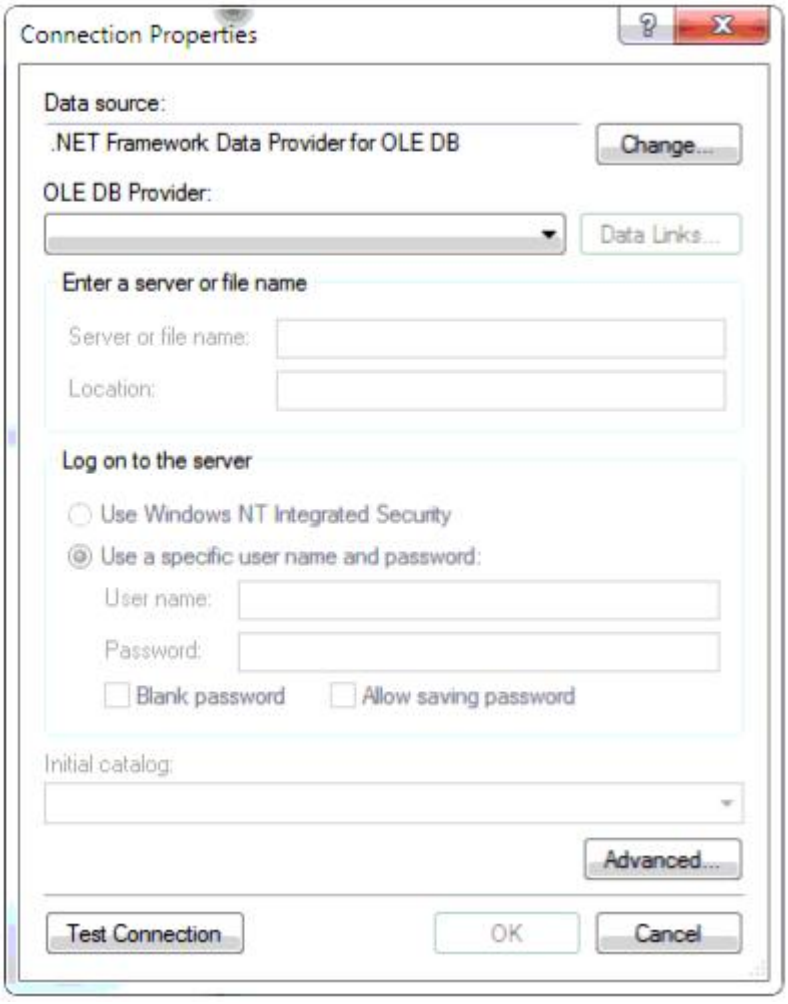
- **Table/Feature Class** (list) – This pane is located along the left side of the form and lists the tables/feature classes that are contained within the data source. Use the check boxes (along the left side of the list) to specify the tables you would like to include.

Note: The list can be resized using the split bar (located on the right side of the list).

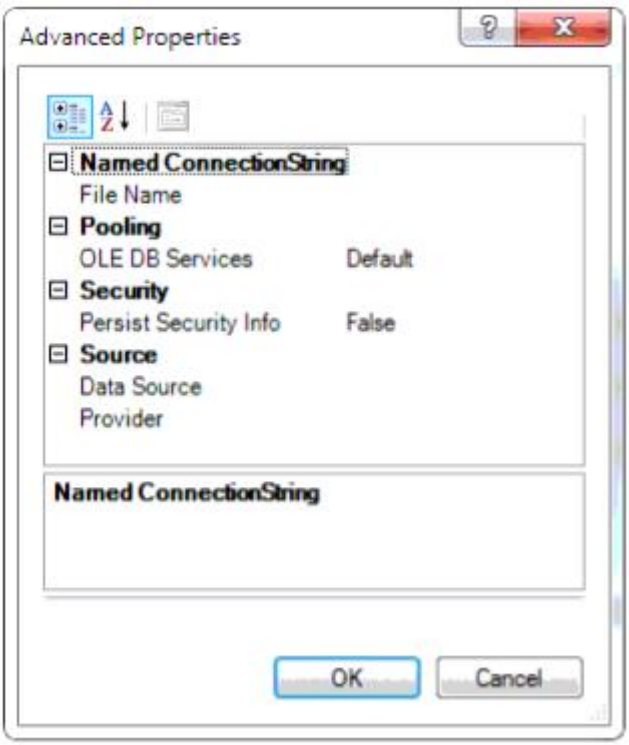
Right-click to Select All or Clear the current selection in the list.

ModelBuilder has built in support for ArcGIS Subtypes. For more information, see ESRI ArcGIS Geodatabase Support.

Connection Properties for OLEDB Source Type
Based on the data source, select the **OLEDB Provider**. For example, **Microsoft Jet 4.0 OLE DB Provider**. The **Data Links** button will provide further settings options. Depending on the OLE DB Provider selected, you may or may not have to provide some/all information. When **Microsoft Jet 4.0 OLE DB Provider** is selected, only **Server or file name** and **User name/Password** are required.



Advanced Connection Properties for OLEDB
Advanced Properties are dependent on the **OLE DB** provider selection. The following screen capture is without selecting any OLE DB provider.

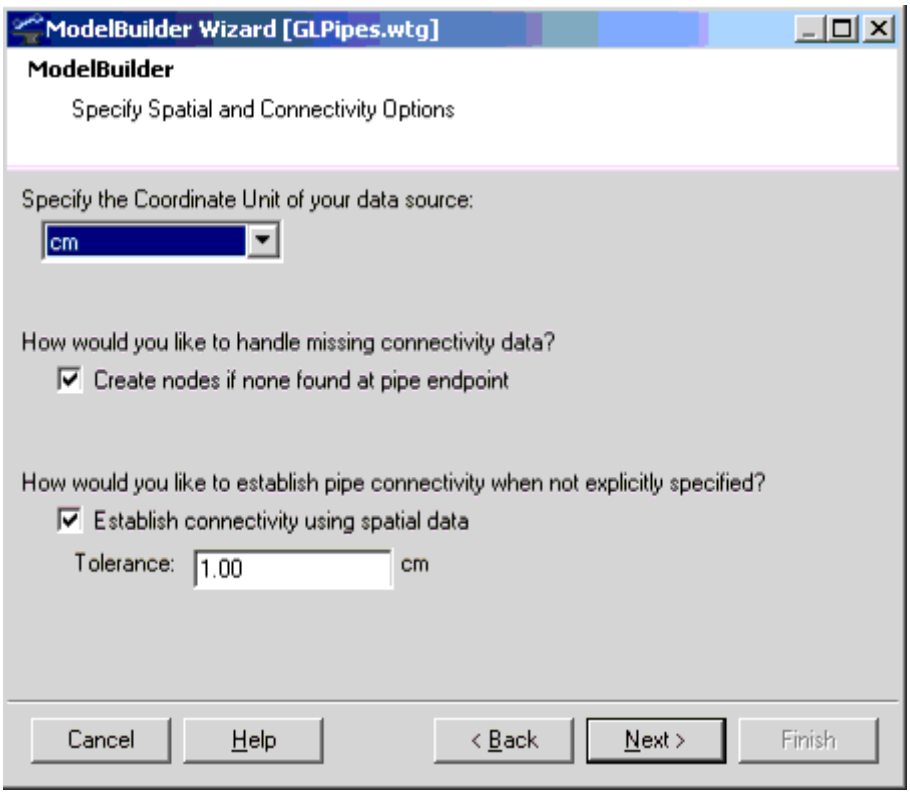


- **Duplicate Table** (button) -- The duplicate table button is located along the top of the Table/Feature Class list. This button allows you to make copies of a table, which can each be mapped to a different element type in your model. Use this in conjunction with the WHERE clause.
- **Remove Table** (button) -- The remove table button can be used to remove a table from the list.
- **WHERE** Clause (field) – Allows you to create a SQL query to filter the records. When the box is checked, only records that meet the criteria specified by the WHERE clause will be displayed. Click the button to validate the query and to refresh the preview table.
- **Preview Pane** – A tabular preview of the highlighted table is displayed in this pane when the Show Preview check box is enabled.

Note: If both nodes and pipes are imported in the same ModelBuilder connection, nodes will be imported first regardless of the order they are listed here.

5.4.2 Step 2 – Specify Spatial Options

In this step you will specify the spatial options to be used during the ModelBuilder process. The spatial options will determine the placement and connectivity of the model elements. The fields available in this step will vary depending on the data source type. It is essential to select or import the correct coordinate system/spatial reference, which should populate the correct X/Y extents.



- **Specify the Coordinate Unit of your data source** (drop-down list) – This field allows you to specify the coordinate unit of the spatial data in your data source. The default unit is the unit used for coordinates.

- **Create nodes if none found at pipe endpoint** (check box) – When this box is checked, ModelBuilder will create a pressure junction at any pipe endpoint that: a) does not have a connected node, and b) is not within the specified tolerance of an existing node. This field is only active when the "Establish connectivity using spatial data" box is checked. (This option is not available if the connection is bringing in only point type geometric data.)

  ModelBuilder will not create pipes unless a valid start/stop node exists. Choose this option if you know that there are nodes missing from your source data. If you expect your data to be complete, then leave this option off; if this situation is detected ModelBuilder will report errors for your review. For more information see the Water GEMS V8i User's Guide

- **Establish connectivity using spatial data** (check box) – When this box is checked, ModelBuilder will connect pipes to nodes that fall within a specified tolerance of a pipe endpoint. (This option is available if the connection is bringing in only polyline type geometric data.) Use this option when the data source does not explicitly name the nodes at the end of each pipe. For more information, see the Water GEMS V8i User's Guide.

- **Tolerance** (numeric field) – This field dictates how close a node must be to a pipe endpoint in order for connectivity to be established. The Tolerance field is only available when the Establish connectivity using spatial data box is checked. (This option is available if the connection is bringing in only polyline type geometric data.) Tolerances should be set as low as possible so that unintended connections are not made. If you are not sure what tolerance to use, try doing

some test runs. Use the Network Review queries to evaluate the success of each trial import. (Be aware that larger tolerance may connect too many pipes while too small a tolerance may miss connections.)

About ProjectWise Geospatial

ProjectWise Geospatial gives spatial context to Municipal Products Group product projects in their original form. An interactive map-based interface allows users to navigate and retrieve content based upon location. The environment includes integrated map management, dynamic coordinate system support, and spatial indexing tools.

ProjectWise Geospatial supports the creation of named spatial reference systems (SRSs) for 2D or 3D Cartesian coordinate systems, automatic transformations between SRSs, creation of Open GIS format geometries, definition of spatial locations, association of documents and folders with spatial locations, and the definition of spatial criteria for document searching.

A *spatial location* is the combination of a geometry for a project plus a designated SRS. It provides a universal mechanism for graphically relating ProjectWise documents and folders.

The ProjectWise administrator can assign background maps to folders, against which the contained documents or projects will be registered and displayed. For documents such as Municipal Products Group product projects, ProjectWise Geospatial can automatically retrieve the embedded spatial location. For documents that are nonspatial, the document can simply inherit the location of the folder into which it is inserted, or users can explicitly assign a location, either by typing in coordinates, or by drawing them.

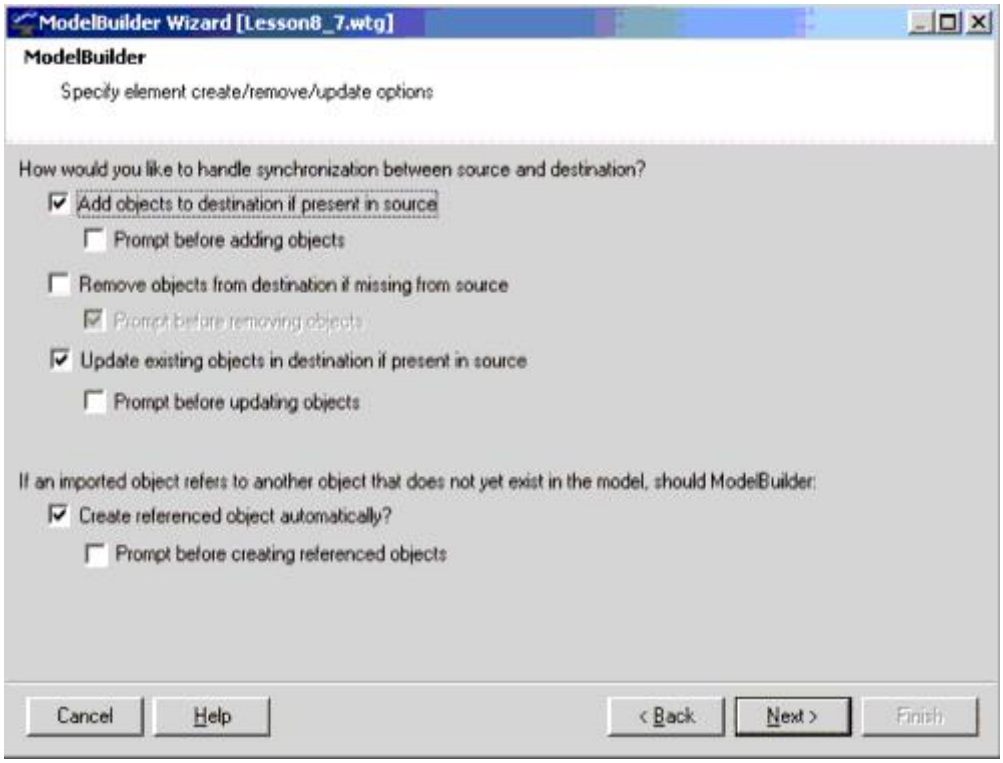
Each document is indexed to a universal coordinate system or SRS, however, the originating coordinate system of each document is also preserved. This enables search of documents across the boundary of different geographic, coordinate, or engineering coordinate systems.

Custom geospatial views can be defined to display documents with symbology mapped to arbitrary document properties such as author, time, and workflow state.

Note: Pipes will be connected to the closest node within the specified tolerance. The unit associated with the tolerance is dictated by the Specify the Coordinate Unit of your data source field.
The standalone version of WaterGEMS does not easily support coordinate systems, but ArcGIS version does.

5.4.3 Step 3 – Specify Element Create/Remove/Update Options

Because of the variety of different data sources and the way those sources were created, the user has a wide variety of options to control the behavior of ModelBuilder.



How would you like to handle synchronization between source and destination?

- **Add objects to destination if present in source** (check box) – When this box is checked, ModelBuilder will automatically add new elements to the model for "new" records in the data source when synching in (or vice-versa when synching out).

  This is checked by default since a user generally wants to add elements to the model (especially if this is the initial run of ModelBuilder). This should be unchecked if new elements have been added to the source file since the model was created but the user does not want them in the model (e.g. proposed piping).

  - ✓ **Prompt before adding objects** (check box) – When this box is checked, ModelBuilder will pause during the synchronization process to present a confirmation message box to the user each time an element is about to be created in the model or data-source.

- **Remove objects from destination if missing from source (**check box) – When this box is checked, ModelBuilder will delete elements from the model if they do not exist in the data source when synching in (or vice-versa when synching out). This option can be useful if you are importing a subset of elements.

  This is used if abandoned pipes have been deleted from the source file and the user wants them to be removed automatically from the model by ModelBuilder.

  - ✓ **Prompt before removing objects** (check box) – When this box is checked, ModelBuilder will pause during the synchronization process to present a

confirmation message box to the user each time an element is about to be deleted from the model.**(Be aware this option must be used with care.)**

- **Update existing objects in destination if present in source** (check box) – If checked, this option allows you to control whether or not properties and geometry of existing model elements will be updated when synching in (or vice-versa when synching out). Turning this option off can be useful if you want to synchronize newly added or removed elements, while leaving existing elements untouched.(This is very useful when adding attributes.)

  - ✓ **Prompt before updating objects** (check box) – When this box is checked, ModelBuilder will pause during the synchronization process to present a confirmation message box to the user each time an element is about to be updated.

If an imported object refers to another object that does not yet exist in the model, should ModelBuilder:

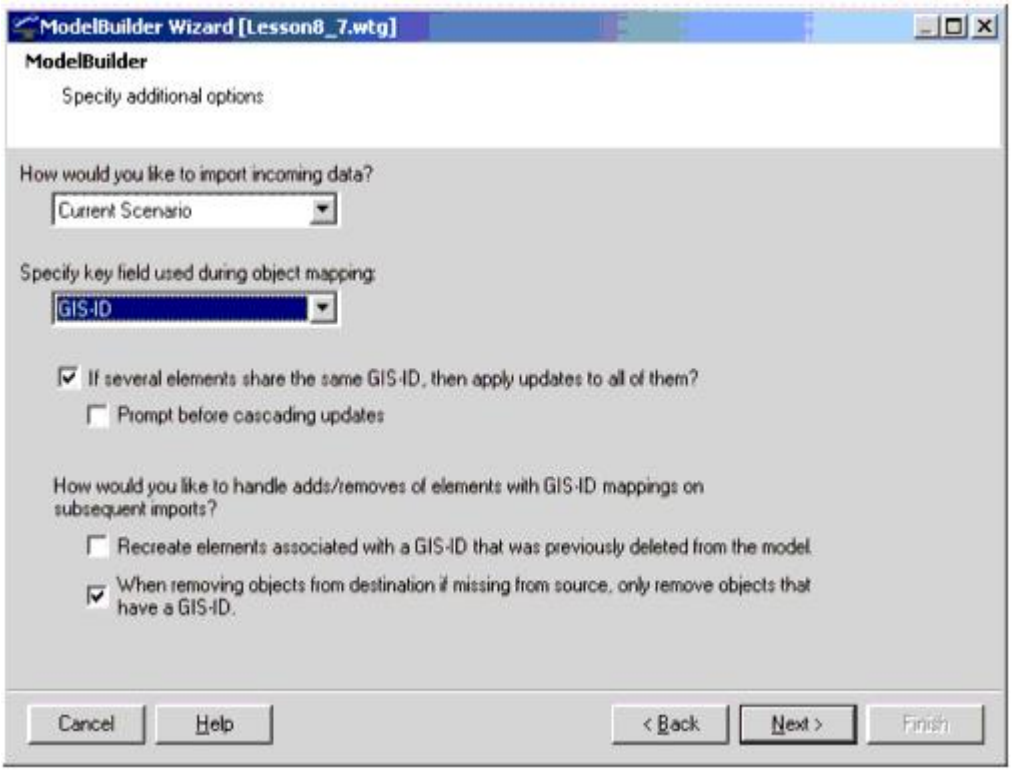
- **Create referenced element automatically**? (check box) – When this box is checked, ModelBuilder will create any domain and/or support elements that are referenced during the import process.

  - ✓ **Prompt before creating referenced elements** (check box) –When this box is checked, ModelBuilder will pause during model generation to present a confirmation message box to the user each time a specified referenced element could not be found and is about to be created for the model.

    "Referenced elements" refers to any support or domain element that is referenced by another element. For example, Pumps can refer to Pump Definition support-elements, Junctions can refer to Zone support-elements, and Pumps can refer to a downstream Pipe domain-element. Node domain-elements that get created as a result of being referenced during the ModelBuilder process will use a default coordinate of 0, 0.

Note: Options listed above apply to elements (pipes and nodes) as well as support elements (such as Zones or Controls).

5.4.4 Step 4 – Additional Options



- **How would you like to import incoming data?** (drop-down list) – This refers to the scenario (and associated alternatives) into which the data will be imported. The user can import the data into the Current Scenario or a new child scenario. If the latter is selected, a new child scenario (and child alternatives) will be created for any data difference between the source and the active scenario. The Base option can be used to import data in all scenarios in the model.

  Note: If there is no data change for a particular alternative, no child alternative will be created in that case.

  New scenario and alternatives will be automatically labeled "Created by ModelBuilder" followed by the date and time when they were created.

- **Specify key field used during object mapping** (drop-down list) –The key field represents the field in the model and data source that contains the unique identifier for associating elements in your model to records in your data source. Refer to the "Key Field (Model)" topic in the next section for additional guidance on how this setting applies to ModelBuilder. ModelBuilder provides three choices for Key Field:

  - ✓ **Label** - The element "Label" will be used as the key for associating model elements with data source records. Label is a good choice if the identifier field in your data-source is unique and represents the identifier you commonly use to refer to the record in your GIS. (Be aware this option is not acceptable if there are duplicate labels)

  - ✓ **<custom>** - Any editable text field in your model can be used as the key for associating model elements with data source records. This is a good choice if you don't use labels on every element, or if perhaps there are duplicate labels in your data source.

- ✓ **GIS-ID** - The element "GIS-ID" field will be used as the key for associating model elements with data source elements. The GIS-ID field offers a number of advanced capabilities and is the preferred choice for models that you plan to keep in sync with your GIS over a period of time.

  Refer to the section Property for more information.

The following options apply only when using the advanced GIS-ID key field option.

- **If several elements share the same GIS-ID, then apply updates to all of them?**(check box)–When using the GIS-ID option, ModelBuilder allows you to maintain one-to-many, and many-to-one relationships between records in your GIS and elements in your Model.

  For example, you may have a single pipe in your GIS that you want to maintain as multiple elements in your Model because you have split that pipe into two pipe elements in the model. You may accomplish this using the native WaterGEMS V8i layout tools to split the pipe with a node; the newly created pipe segment will be assigned the same GIS-ID as the original pipe (establishing a one-to-many relationship). By using this option, when you later synchronize from the GIS into your model, any data changes to the single pipe record in your GIS can be cascaded to both pipes elements in your model (e.g. so a diameter change to a single record in the GIS would be reflected in both elements in the model).If a GIS pipe is split into several model pipes, note that the pipe length should not be updated in the future.

  - ✓ **Prompt before cascading updates** (check box) – When this box is checked, ModelBuilder will pause during model generation to present a confirmation message box to the user each time a cascading update is about to be applied.

- **How would you like to handle add/removes of elements with GIS-ID mappings on subsequent imports?** These options are useful for keeping your GIS and Model synchronized, while maintaining established differences.

  - **Recreate elements associated with a GIS-ID that was previously deleted from the model** (check box) – By default, ModelBuilder will not recreate elements you remove from your model that are associated with records (with GIS-ID mappings) that are still in your GIS. This behavior is useful when you want to perform GIS to model synchronizations but have elements in your GIS that you do not want in your model.

    For example, after creating your model from GIS, you may find redundant nodes when performing a Network Navigator, "Nodes in Close Proximity" network review query. You may choose to use the "Merge Nodes in Close Proximity" feature to make the correction in your model (deleting the redundant nodes from your model). Normally, when you later synchronize from your GIS to your model, missing elements would be recreated and your correction would be lost.

    However, WaterGEMS V8i now maintains the history of elements (with GIS-ID's) that were removed from your model; this option allows you to control whether or not those elements get recreated.

- **When removing objects from destination if missing from source, only remove objects that have a GIS-ID (check box)** – This option is useful when you have

elements that are missing from your GIS that you want to keep in your model (or vice-versa).

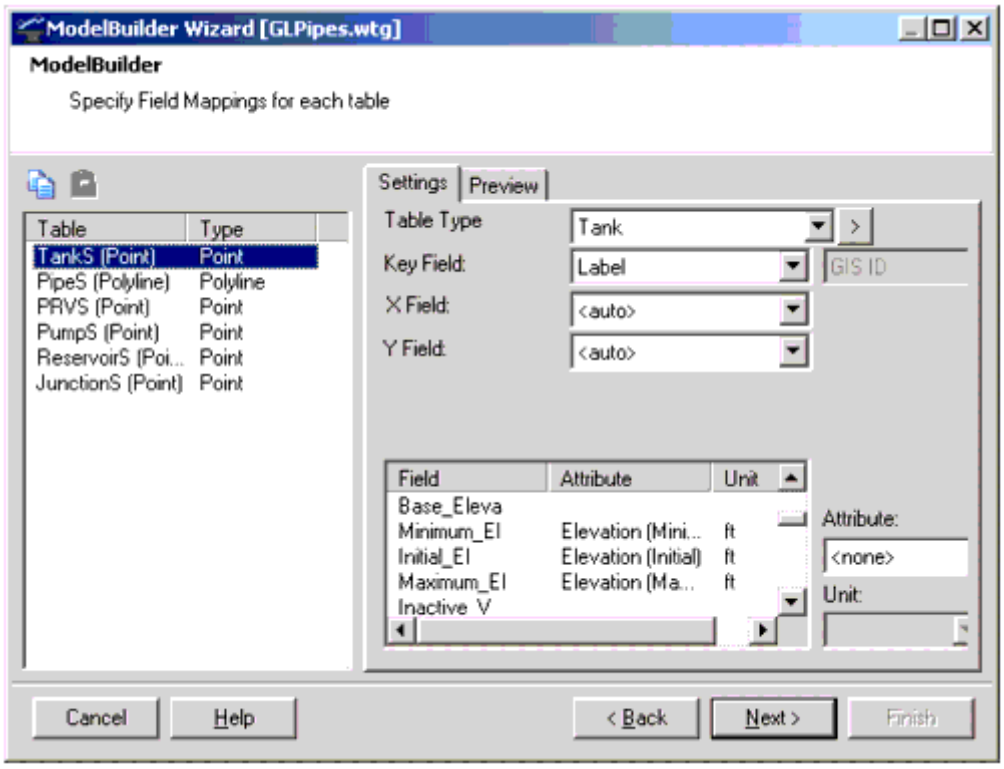
For example, if you build your model from your GIS (using the GIS-ID option, a GIS-ID will be assigned to newly created elements in your model. If you later add elements to your model (they will not be assigned a GIS-ID); on subsequent synchronizations, this option (if checked) will allow you to you retain those model specific elements that do not exist in your GIS. For example, you may have a proposed land development project in your model that does not exist in the GIS. These elements will not have a GIS-ID because they were not imported from the GIS. If this box is checked, the new elements will not be removed on subsequent runs of ModelBuilder.

Note: *This setting only applies if the "Remove objects from destination if missing from source" option is checked.*

When you do make connectivity changes to your model, it is often beneficial to make those same changes to the GIS. However, this is not always possible. In some cases it is not actually desirable, given the fact that modeling often has highly specialized needs that may not be met by a general purpose GIS. It is also suggested that a User Data Extension be set up to keep track of any changes made to model data (e.g., name of person modifying data, date, and reason)

5.4.5 Step 5 – Specify Field mappings for each Table/Feature Class

In this step, data source tables are mapped to the desired modeling element types, and data source fields are mapped to the desired model input properties. You will assign mappings for each Table/Feature Class that appears in the list. Step 1 of the wizard can be used to exclude tables, if you wish.



- **Tables** (list) – This pane, located along the left side of the dialog box, lists the data source Tables/Feature Classes to be used in the ModelBuilder process. Select an item in the list to specify the settings for that item.

Note: The tables list can be resized using the splitter bar.

There are two toolbar buttons located directly above Tables list (these buttons can be a great time saver when setting up multiple mappings with similar settings).

- ✓ **Copy Mappings** (button) – This button copies the mappings (associated with the currently selected table) to the clipboard.

- ✓ **Paste Mappings** (button) – This button applies the copied mappings to the currently selected table.

- **Settings Tab** – The Settings tab allows you to specify mappings for the selected item in the Tables list.

The top section of the Settings tab allows you to specify the common data mappings:

- ✓ **Table Type** (drop-down list) – This field, which contains a list of all of the WaterGEMS V8i/Hammer element types, allows you to specify the target modeling element type that the source table/feature class represents. For example, a source table that contains pipe data should be associated with the Pressure Pipe element type.

  There are three categories of Table Types: Element Types, Components, and Collections. For geometric data sources, only Element Types are available. However with tabular data sources all table types can be used. The categorized menu accessed by the [>] button assists in quicker selection of the desired table type.

  - **Element Types** – This category of Table Type includes geometric elements represented in the drawing view such as pipes, junctions, and tanks.
  - **Components** – This category of Table Type includes the supporting data items in your model that are potentially shared among elements such as patterns, pump definitions, and controls.
  - **Collections** – This category of Table Type includes table types that are typically lists of two-columned data. For instance, if one table in your connection consists of a list of (Time from Start, Multiplier) pairs, use a Pattern collection table type selection.

  When you are importing a collection, values will always override existing collection items in the model. In order to preserve existing items, they need to be combined with the new values and imported together. For example importing "Junction, Demand Collection", incoming demand rows will override the existing demand collection, not append to it. If you want to keep the existing demands, you should first export those values (copy-paste is usually easiest) to your data source (e.g. spreadsheet, shapefile) and make those demands part of the data you are importing. In this way ModelBuilder will import both the original and new demands.

- ✓ **Key Fields** - This pair of key fields allows you to control how records in your data source are associated with elements in the model. The Key Fields element mapping consists of two parts, a data-source part and a model part:

- **Key Field (Data Source)** (drop-down list) – Choose the field in your data source that contains the unique identifier for each record.

  Note: If you plan to maintain synchronizations between your model and GIS, it is best to define a unique identifier in your data source for this purpose. Using an identifier that is unique across all tables is critical if you wish to maintain explicit pipe start/stop connectivity identifiers in your GIS.

  When working with ArcGIS data sources, OBJECTID is not a good choice for Key field (because OBJECTID is only unique for a particular Feature Class).

  For one-time model builds – if you do not have a field that can be used to uniquely identify each element, you may use the <label> field (which is automatically generated by ModelBuilder for this purpose).

- **Key Field (Model)** (drop-down-list) – This field is only enabled if you specified <custom> in the "Specify key field to be used in object mapping?" option in the previous step. If you specified "GIS-ID' or "Label" the field will be disabled.

If you specified <custom>, then you will be presented with a list of the available text fields for that element type. Choose a field that represents the unique alphanumeric identifier for each element in your model.

Note: You can define a text User Data Extensions attribute for use as your <custom> model key field.

The <custom> key field list is limited to read-write text fields. This is because during import, the value of this field will be assigned as new elements in your model are created. Therefore, the model's internal (read-only) element ID field cannot be used for this purpose.

The following optional fields are available for Pipe element types:

- **Start/Stop** - Select the fields in a pipe table that contain the identifier of the start and stop nodes. Specify <none> if you are using the spatial connectivity support in ModelBuilder (or if you want to keep connectivity unchanged on update). For more information, see Specifying Network Connectivity in ModelBuilder.

  Note: When working with an ArcGIS Geometric Network data source, these fields will be set to <auto> (indicating that ModelBuilder will automatically determine connectivity from the geometric network).

These fields are available for Node element types:

- **X/Y Field** - These fields are used to specify the node X and Y coordinate data. This field only applies to point table types.

Note: The Coordinate Unit setting in Step 2 of the wizard allows you to specify the units associated with these fields.

When working with ArcGIS Geodatabase, shape file and CAD data sources, these fields will be set to <auto> (indicating that ModelBuilder will automatically determine node geometry from the data source).

These optional fields are available for Pump element types:

- **Suction Element** (drop-down list) – For tables that define pump data, select a pipe label or other unique identifier to set the suction element of the Pump.
- **Downstream Edge** (drop-down list) – For tables that define pump or valve data, select a pipe label or other unique identifier to set the direction of the pump or valve.

  The bottom section of the Settings tab allows you to specify additional data mappings for each field in the source.

- **Field** –Field refers to a field in the selected data source. The Field list displays the associations between fields in the database to properties in the model.
- **Attribute** (drop-down list) – Attribute refers to a Bentley WaterGEMS V8i Attribute. Use the Attribute drop-down list to map the highlighted field to the desired Attribute.
- **Unit** (drop-down list) – This field allows you to specify the units of the values in the database (no conversion on your part is required). This field only applies if the selected model property is unitized.

- **Preview Tab** – The Preview tab displays a tabular preview of the currently highlighted source data table when the Show Preview check box is checked.

To map a field in your table to a particular Bentley WaterGEMS V8i property:

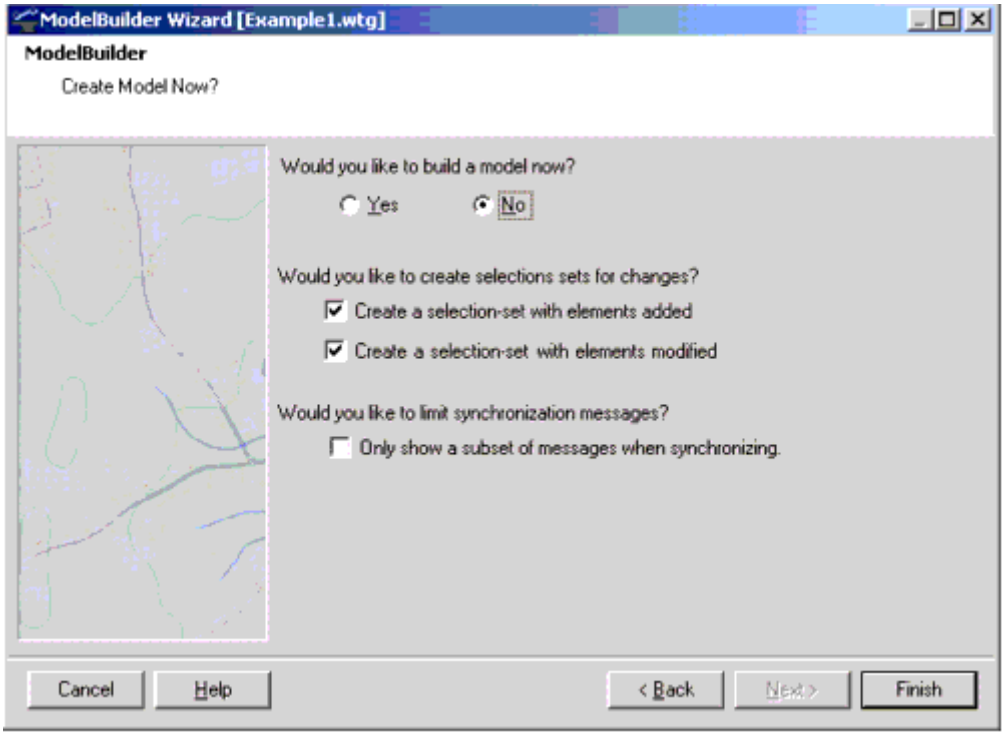
1. In the Field list, select the data source field you would like to define a mapping for.
2. In the Property drop-down list, select the desired Bentley WaterGEMS V8i target model property.
3. If the property is unitized, specify the unit of this field in your data source in the Unit drop-down list.

To remove the mapping for a particular field:

1. Select the field you would like to update.
2. In the Property drop-down list, select <none>.

5.4.6 Step 6 – Build Operation Confirmation

In this step, you are prompted to build a new model or update an existing model.



To build a new model, click the **Yes** radio button under **Would you like to build the model now?**

If you choose **No**, you will be returned to the **ModelBuilder Manager** dialog. The connection you defined will appear in the list pane. To build the model from the ModelBuilder Manager, highlight the connection and click the **Build Model** button.

Create Selection Set Options: Often a user wants to view the elements that have been affected by a ModelBuilder operation. To do this, ModelBuilder can create selection sets which the user can view and use within the application.

- To create a selection set containing the elements added during the ModelBuilder, check the box next to **"Create selection set with elements added."**
- To create a selection set containing the elements for which the properties or geometry were modified during the ModelBuilder, check the box next to **"Create selection set with elements modified."**

Only show a subset of messages when synchronizing: Depending on the ModelBuilder configuration and the external data, there are situations when a very large number of messages may be generated during the ModelBuilder synchronization. Generating these messages adds some overhead and can use up a large amount of memory. Checking this box will limit the number of messages that are generated for each specific message type.

Note: Selection sets created as a result of these options will include the word "ModelBuilder" in their name, along with the date and time (e.g. "Elements added via ModelBuilder - mm/dd/yyyy hh:mm:ss am/pm")

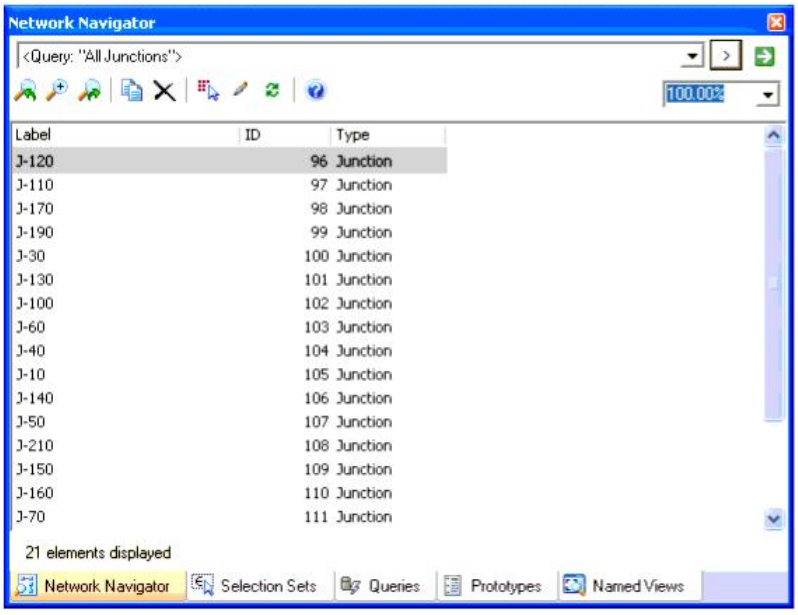
5.5 Reviewing Your Results

At the end of the model building process, you will be presented with statistics, and a list of any warning/error messages reported during the process. You should closely review this information, and be sure to save this data to disk where you can refer to it later.

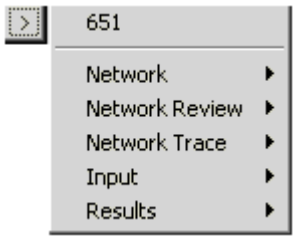
Refer to the Using the Network Navigator and Manipulating Elements topics for information about reviewing and correcting model connectivity issues.

Using the Network Navigator
The Network Navigator consists of a toolbar and a table that lists the Label and ID of each of the elements contained within the current selection. The selection can include elements highlighted manually in the drawing pane, elements contained within a selection set, or elements returned by a query.

To open the Network Navigator, click the View menu and select the Network Navigator command, press <Ctrl+3>, or click the Network Navigator button  on the View toolbar.



The following controls are included in Network Navigator:



Network Review
Network Review Queries include the following:

- **Nodes In Close Proximity** - Identifies nodes within a specific tolerance.
- **Crossing Pipes** - Identifies pipes that intersect one another with no junction at the intersection.

- **Orphaned Nodes** - Identifies nodes that are not connected to a pipe in the model.
- **Orphaned Isolation Valves** - Identifies isolation valves that are not connected to a pipe in the model.
- **Dead End Nodes** - Identifies nodes that are only connected to one pipe.
- **Dead End Junctions** - Identifies junctions that are only connected to one pipe.
- **Pipe Split Candidates**- Identifies nodes near a pipe that may be intended to be nodes along the pipe. The tolerance value can be set for the maximum distance from the pipe when the node should be considered as a pipe split candidate.
- **Pipes Missing Nodes** - Identifies which pipes are missing either one or both end nodes.
- **Duplicate Pipes** - Identifies instances in the model where a pipe shares both end nodes with another pipe.

Network Trace
Network Trace Queries include the following:

- **Find Connected** - Locates all the connected elements to the selected element in the network.
- **Find Adjacent Nodes** - Locates all node elements connected upstream or downstream of the selected element or elements.
- **Find Adjacent Links** - Locates all link elements connected upstream or downstream of the selected element or elements.
- **Find Disconnected** - Locates all the disconnected elements in the network by reporting all the elements not connected to the selected element.
- **Find Shortest Path** - Select a Start Node and a Stop Node. The query reports the shortest path between the two nodes based upon the shortest number of edges.
- **Trace Upstream** - Locates all the elements connected upstream of the selected downstream element.
- **Trace Downstream** - Locates all the elements connected downstream of the selected upstream element.
- **Isolate** - Selects an element that needs to be serviced. Run the query to locate the nearest isolation valves. In order to service the element, this will identify where shut off points and isolation valves are located.
- **Find Initially Isolated Elements** - Locates elements that are not connected or cannot be reached from any boundary condition.

6 THE IMPORTANCE OF ACCURATE ELEVATION DATA

Obtaining node elevation data for input into a water distribution model can be an expensive, time-consuming process. In some cases, very accurate elevation data may be critical to the model's utility; in other cases it can represent a significant resource expenditure. In order to decide on the appropriate level of quality of elevation data to be gathered, it is important to understand how a model uses this data.

Elevation data for nodes is not directly used in solving the network equations in hydraulic models. Instead, the models solve for hydraulic grade line (HGL). Once the HGL is calculated and the numerical solution process is essentially completed, the elevations are then used to determine pressure using the following relationship:

$$p = (\mathrm{HGL} - z)\rho g$$

Where: p = pressure (lb./ft.^{2}, N/m^2)
HGL = hydraulic grade line (ft., m)
z = node elevation (ft., m)
p = density of water (slugs/ft.^{3}, kg/m^3)
g = gravitational acceleration (ft./sec.^{2}, m/sec.^{2})

If the modeler is only interested in calculating flows, velocities, and HGL values, then elevation need not be specified. In this case, the pressures at the nodes will be computed assuming an elevation of zero, thus resulting in pressures relative to a zero elevation.

If the modeler specifies pump controls or pressure valve settings in pressure units, then the model needs to compute pressures relative to the elevation of the nodes being tested. In this case, the elevation at the control node or valve would need to be specified (or else the model will assume zero elevation). Therefore, an accurate elevation value is required at each key node where pressure is of importance.

6.1 TRex Terrain Extractor

The TRex Terrain Extractor was designed to expedite the elevation assignment process by automatically assigning elevations to the model features according to the elevation data stored within Digital Elevation Models.

Digital Elevation Models were chosen because of their wide availability and since a reasonable level of accuracy can be obtained by using this data type depending on the accuracy of the DEM/DTM.

The TRex Terrain Extractor can quickly and easily assign elevations to any or all of the nodes in the water distribution model. All that is required is a valid Digital Elevation Model. Data input for TRex consists of:

1. Specify the GIS layer that contains the DEM from which elevation data will be extracted.
2. Specify the measurement unit associated with the DEM (feet, meters, etc.).
3. Select the model features to which elevations should be applied; all model features or a selection set of features can be chosen.

TRex then interpolates an elevation value for each specific point occupied by a model feature. The final step of the wizard displays a list of all features to which an elevation was applied, along with the elevation values for those features. These elevation values can then be applied to a new physical properties alternative, or an existing one. In some cases, you might have more accurate information for some nodes (e.g., survey elevation from a pump station). In those cases, you should create the elevation data using DEM data and manually overwrite the more accurate data for those nodes.

The TRex Terrain Extractor simplifies the process of applying accurate elevation data to water distribution models. As was shown previously, accurate elevation data are vital when accurate pressure calculations and/or pressure-based controls are required for the water distribution model in question. All elevation data for even large distribution networks can be applied by completing a few steps.

In the US, DEM data is usually available in files corresponding to a single USGS 7.5 minute quadrangle map. If the model covers an area involving several maps, it is best to mosaic the maps into a single map using the appropriate GIS functions as opposed to applying TRex separately for each map.(Elevations can be assigned one map at a time.)

When using TRex, it is necessary that the model and the DEM be in the same coordinate system. Usually the USGS DEMs are in the UTM (Universal Transverse Mercator) with North American Datum 1983 (NAD83) in meters, although some may use NAD27. Models are often constructed using a state plane coordinate system in feet. Either the model or DEM must be converted so that the two are in the same coordinate system for TRex to work. Similarly, the vertical datum for USGS is based on national Vertical Geodetic Datum of 1929. If the utility has used some other datum for vertical control, then these differences need to be reconciled.

The TRex Terrain Extractor can read the USGS DEM raster data in SDTS (The Spatial Data Transfer Standard) format. Raster profiles provide a flexible way to encode raster data. The SDTS standard contains small limited subsets called profiles. In a raster transfer, there should be one RSDF(The Raster Definition) module, one LDEF(Layer Definition) module and one or more cell modules. Each record in the RSDF module denotes one raster object. Each raster object can have multiple layers. Each layer is encoded as one record in the LDEF module. The actual grid data is stored in the cell module which is referenced by the layer record. A typical USGS DEM data set contains one RSDF record, one LDEF record and one cell file.

Many of the functions that TREX performs can be done in ArcGIS with Spatial Joins and/or Spatial Analysis tools which is very reliable.

6.2 TRex Wizard

The TRex Wizard steps you through the process of automatically assigning elevations to specified nodes based on data from a Digital Elevation Model or a Digital Terrain Model. TRex can load elevation data into model point features (nodes) from a variety of file types, including both vector and raster files. To use raster files as the data source, the ArcGIS platform must be used. With a vector data source, it is possible to use any platform. Vector data must consist of either points with an elevation or contours with an elevation.

It is important to understand the resolution, projection, datum, units and accuracy of any source file that will be used to load elevation data for nodes.

In the United States, elevation data can be obtained at the USGS National Map Seamless Server. The vertical accuracy may only be +/- 7 to 15 m.

6.2.1 Step 1: File Selection

The elevation data source and features to which elevations will be assigned are specified in the File Selection dialog of the TRex wizard. Valid elevation data sources include:

- Vector files such as DXF and SHP files
- LandXML files
- InRoads .dtm (Microstation platform only)
- Geopack .tin (32-bit version only)
- Bentley MX .fil
- Bentley .dgn (Microstation platform only)

DXF files are able to contain both points and lines; the user must therefore indicate whether the node elevations should be built based on the points in the DXF, or based on the contour lines in the DXF.

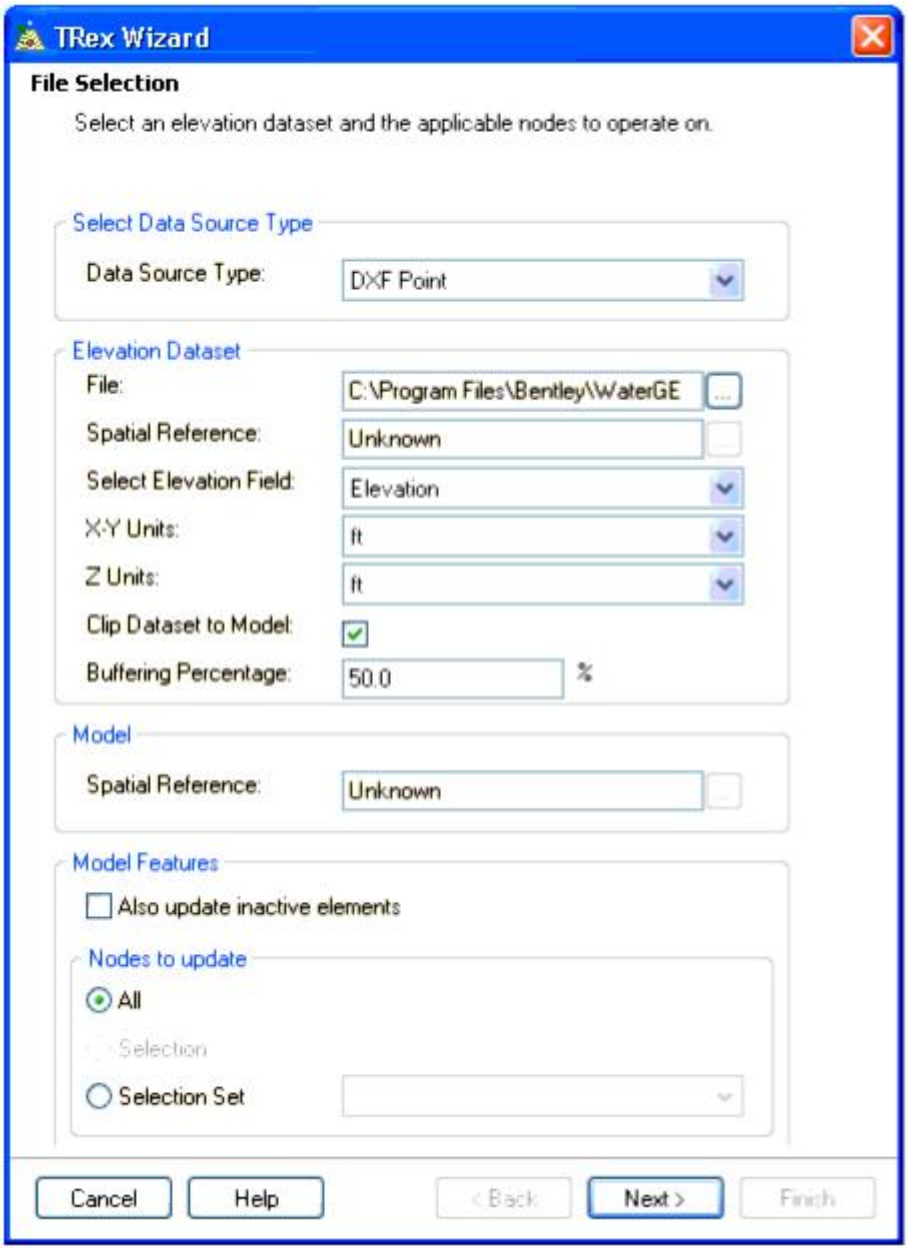
Shapefiles are not allowed to contain mixed geometric data, so TRex can safely determine whether to build the elevation map based on either elevation point data or elevation contour lines. The Model Spot Elevation data source type uses existing spot elevation nodes in the model, which must already have correct elevation values assigned. Using these as the data source, TRex can determine the elevations for the other nodes in the model.

Bentley MX (.fil) files can contain multiple terrain models; you must select a single model to use as the elevation data source.

When running under the ArcGIS platform, additional raster data sources are also available for direct use in TRex, including TIN, Rasters(grid), USGS(DEM), and SDTS(DDF) files.

These data sources are often created in a specific spatial reference, meaning that the coordinates in the data source will be transformed to a real geographic location using this spatial reference. Care must be taken when laying out the model to ensure that the model coordinates, when transformed by the model's spatial reference (if applicable), will overlay the elevation data source in this 'global' coordinate system. If the model and elevation data source's data do not overlay each other, TRex will be unable to interpolate elevation data. GIS products such as Bentley Map and ArcGIS can be used to transform raster source data into a spatial reference that matches that of the model.

If you are unable to run TRex under ArcGIS (i.e. you are using stand-alone or a CAD platform), ArcGIS can generally be used to convert the raster data to a point shapefile that approximates the raster data source. Shapefiles can be always be used in TRex, regardless of the platform that TRex is running.



- **Data Source Type** – This menu allows you to choose the type of file that contains the input data you will use.

- **File** – This field displays the path where the data file is located. Use the browse button to find and select the desired file.

- **Spatial Reference (ArcGIS Mode Only)** – Click the Ellipsis (...) next to this field to open the Spatial Reference Properties dialog box, allowing you to specify the spatial reference being used by the elevation data file.

- **Select Elevation Field** – Select the elevation unit.

- **X-Y Units** – This menu allows the selection of the measurement unit type associated with the X and Y coordinates of the elevation data file.

- **Z Units** – This menu allows the selection of the measurement unit type associated with the Z coordinates of the elevation data file.

- **Clip Dataset to Model** – In some cases, the data source contains elevation data for an area that exceeds the dimensions of the area being modeled. When this box is checked, TRex will calculate the model's bounding box, find the larger dimension (width or height), calculate the Buffering Percentage of that dimension, and increase both the width and height of the model bounding box by that amount. Then any data point that falls outside of the new bounding box will not be used to generate the elevation mesh. If this box isn't checked, all the source data points are used to generate the elevation mesh. Checking this box should result in faster calculation speed and use less memory.

- **Buffering Percentage** – This field is only active when the Clip Dataset to Model box is checked. The percentage entered here is the percentage of the larger dimension (width or height) of the model's bounding box that will be added to both the bounding box width and height to find the area within which the source data points will be used to build the elevation mesh. (Use of bounding boxes and extrapolating outside the box will lead to errors.)

- **Spatial Reference (ArcGIS Mode Only)** – Click the Ellipsis (...) next to this field to open the Spatial Reference Properties dialog box, allowing you to specify the spatial reference being used by the WaterGEMS V8i model file.

- **Also update inactive elements** – Check this box to include inactive elements in the elevation assignment operation. When this box is unchecked, elements that are marked Inactive will be ignored by TRex.

- **All** – When this button is selected, TRex will attempt to assign elevations to all nodes within the WaterGEMS V8i model.

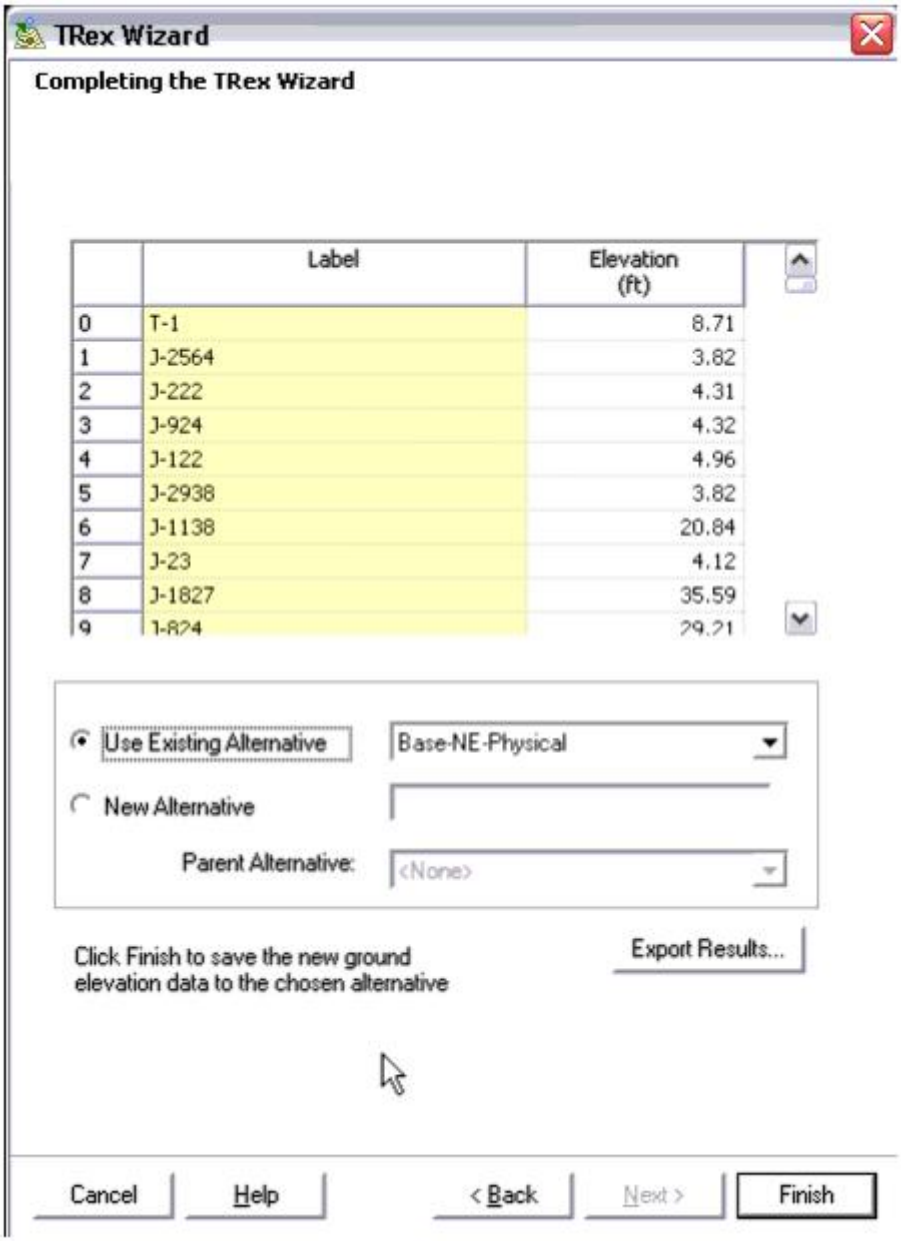
- **Selection** – When this button is selected, TRex will attempt to assign elevations to all currently highlighted nodes.

- **Selection Set** – When this is selected, the Selection Set menu is activated. When the Selection Set button is selected, TRex will assign elevations to all nodes within the selection set that is specified in this menu.

  Note: If the WaterGEMS V8i model (which may or may not have a spatial reference explicitly associated with it) is in a different spatial reference than the DEM/DTM (which does have a spatial reference explicitly associated with it), then the features of the model will be projected from the model's spatial reference to the spatial reference used by the DEM/DTM.

6.2.2 Step 2: Completing the TRex Wizard

The results of the elevation extraction process are displayed and the results can be applied to a new or existing physical alternative.



- **Results Preview Pane** – This tabular pane displays the elevations that were calculated by TRex. The table can be sorted by label by clicking the Label column heading and by elevation by clicking the Elevation column heading. You can filter the table by right-clicking a column in the table and selecting the **Filter...Custom** command. You can also right-click any of the values in the elevation column to change the display options.

- **Use Existing Alternative** – When this is selected, the results will be applied to the physical alternative that is selected in the Use Existing Alternative menu. This menu allows the selection of the physical alternative to which the results will be applied.

- **New Alternative** – When this is selected, the results will be applied to a new physical alternative. First, the currently active physical alternative will be duplicated, and then the results generated by TRex will be applied to the newly created alternative. The name of this new alternative must be supplied in the New Alternative text field.

- **Parent Alternative** – Select an alternative to duplicate from the menu, or select <None> to create a new Base alternative.

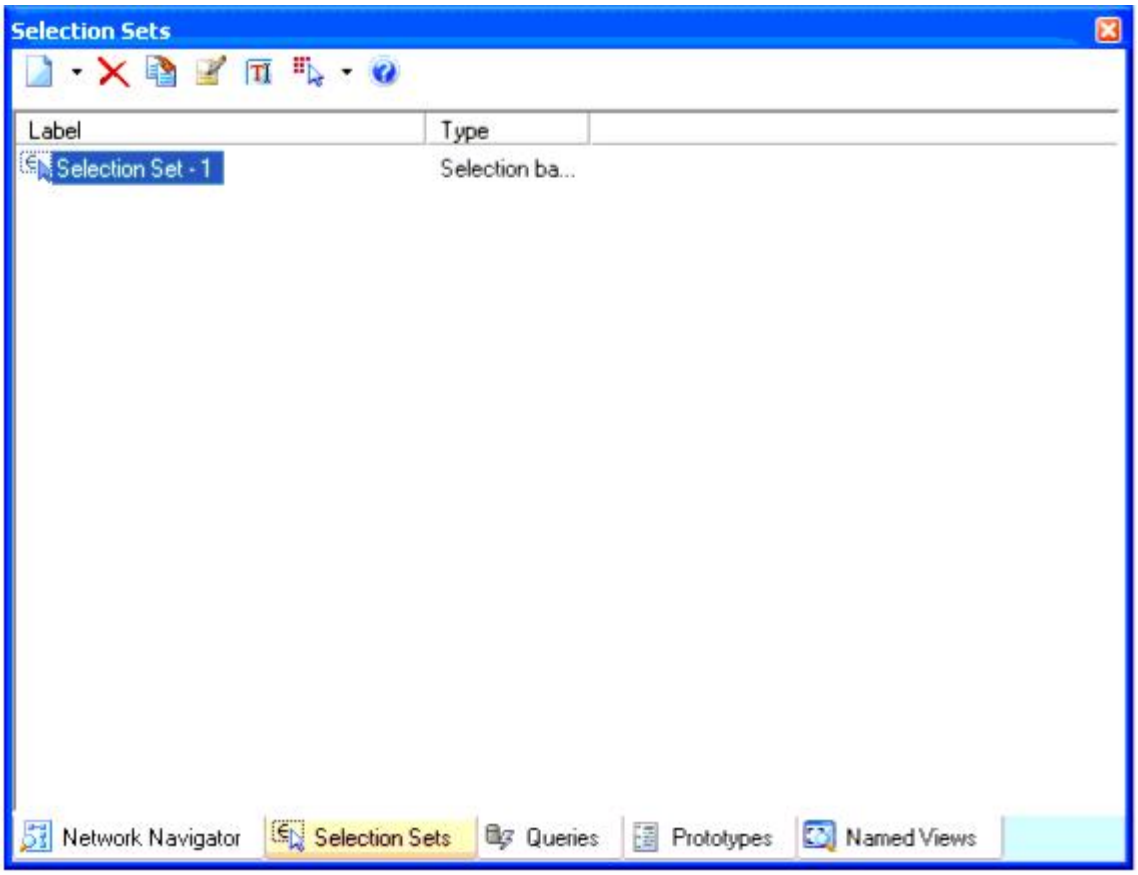
- **Export Results** – This exports the results generated by TRex to a tab or comma-delimited text file (.TXT). These files can then be re-used by WaterGEMS V8i or imported into other programs.

- Click Finish when complete, or Cancel to close without making any changes.

7 SELECTION SETS

The Selection Sets Manager is used to create, edit, and navigate to selection sets. The Selection Sets Manager consists of a toolbar and a list pane, which displays all of the selection sets that are associated with the current project.

To open Selection Sets, click the View menu and select the Selection Sets command, press <Ctrl+4>, or click the Selection Sets button  on the View toolbar.



To view elements in a Selection Set
You use the Network Navigator to view the elements that make up a selection set.

1. Open the Network Navigator by selecting View > Network Navigator or clicking the Network Navigator button on the View toolbar.
2. Select a selection set from the Selection Set drop-down list. The elements in the selection set appear in the Network Navigator.

   Note: You can double-click an element in the Network Navigator to select and center it in the Drawing Pane.

To Create a Selection Set from a Selection
You create a new selection set by selecting elements in your model.

1. Select all of the elements you want in the selection set by either drawing a selection box around them or by holding down the Ctrl key while clicking each one in turn.

2. When all of the desired elements are highlighted, right-click and select **Create Selection Set**.
3. Type the name of the selection set you want to create, and then click OK to create the new selection set. Click **Cancel** to close the dialog box without creating the selection set.
4. Alternatively, you can open the Selection Set manager and click the **New** button and select **Create from Selection**. Bentley WaterGEMS V8i prompts you to select one or more elements.

Create Selection Set Dialog Box
This dialog box opens when you create a new selection set. It contains the following field:

New selection set name	Type the name of the new selection set.

To create a Selection set from a Query
You create a dynamic selection set by creating a query-based selection set. A query-based selection set can contain one or more queries, which are valid SQL expressions.

1. In the Selection Sets Manager, click the **New** button and select **Create from Query**. The Selection by Query dialog box opens.
2. Available queries appear in the list pane on the left; queries selected to be part of the selection set appear in the list pane on the right. Use the arrow buttons in the middle of the dialog to add one or all queries from the Available Queries list to the Selected Queries list, or to remove queries from the Selected list.
    - You can also double-click queries on either side of the dialog box to add them to or remove them from the selection set.

Selection by Query Dialog Box
The Selection by Query dialog box is used to create selection sets from available queries. The dialog box contains the following controls:

Available Queries	Contains all the queries that are available for your selection set. The Available Columns list is located on the left side of the dialog box.
Selected Queries	Contains queries that are part of the selection set. To add queries to the Selected Queries list, select one or more queries in the Available Queries list, then click the **Add** button [>].
Query Manipulation Buttons	Select or clear queries to be used in the selection set: • [>] Adds the selected items from the Available Queries list to the Selected Queries list. • [>>] Adds all of the items in the Available Queries list to the Selected Queries list. • [<] Removes the selected items from the Selected Queries list. • [<<] Removes all items from the Selected Queries list. Note: You can select multiple queries in the Available Queries list by holding down the Shift key or the Control key while clicking with the mouse. Holding down the Shift key provides group selection behavior. Holding down the Control key provides single element selection behavior.

To add elements to a Selection Set
You can add a single or multiple elements to a static selection set.

1. Right-click the element to be added, then select **Add to Selection Set** from the shortcut menu.
2. In the Add to Selection Set dialog box, select the selection set to which you want to add the element.
3. Click **OK** to close the dialog box and add the element to the selected selection set. Click **Cancel** to close the dialog box without creating the selection set

   Note: that previously selected elements can be included in a new selection set if desired

To add a group of elements to a static selection set all at once
1. Select all of the elements to be added by either drawing a selection box around them, or by holding down the **Ctrl** key while clicking each one in turn.
2. When all of the desired elements are highlighted, right-click and select **Add to Selection Set**.
3. In the Add to Selection Set dialog box, select the selection set to which you want to add the element.
4. Click **OK** to close the dialog box and add the element to the selected selection set. Click **Cancel** to close the dialog box without creating the selection set.

To Add To Selection Set Dialog Box
This dialog box opens when you select the Add to Selection Set command. It contains the following field:

Add to:	Selects the selection set to which the currently highlighted element or elements will be added.

To remove elements from a Selection Set
You can easily remove elements from a static selection set in the Selection Set Element Removal dialog box.

1. Display the Selection Sets Manager by selecting **View > Selection Sets** or clicking **the Selection Sets** button on the View toolbar.
2. In the Selection Sets Manager, select the desired selection set then click the **Edit** button.
3. In the Selection Set Element Removal dialog box, find the element you want to remove in the table. Select the element label or the entire table row, and then click the **Delete** button.
4. Click **OK**.

Selection Set Element Removal Dialog Box
This dialog opens when you click the edit button from the Selection Sets manager. It is used to remove elements from the selection set that is highlighted in the **Selection Sets Manager** when the **Edit** button is clicked.

7.1.1 Using Queries
A query in Bentley WaterGEMS V8*i* is a user-defined SQL expression that applies to a single element type. You use the Query Manager to create and store queries; you use the Query Builder dialog box to construct the actual SQL expression.

Queries can be one of the following three types:

- **Project queries**—Queries you define that are available only in the Bentley WaterGEMS V8*i* project in which you define them.
- **Shared queries**—Queries you define that are available in all Bentley WaterGEMS V8*i* projects you create. You can edit shared queries.
- **Predefined queries**—Factory-defined queries included with Bentley WaterGEMS V8*i* that are available in all projects you create. You cannot edit predefined queries.

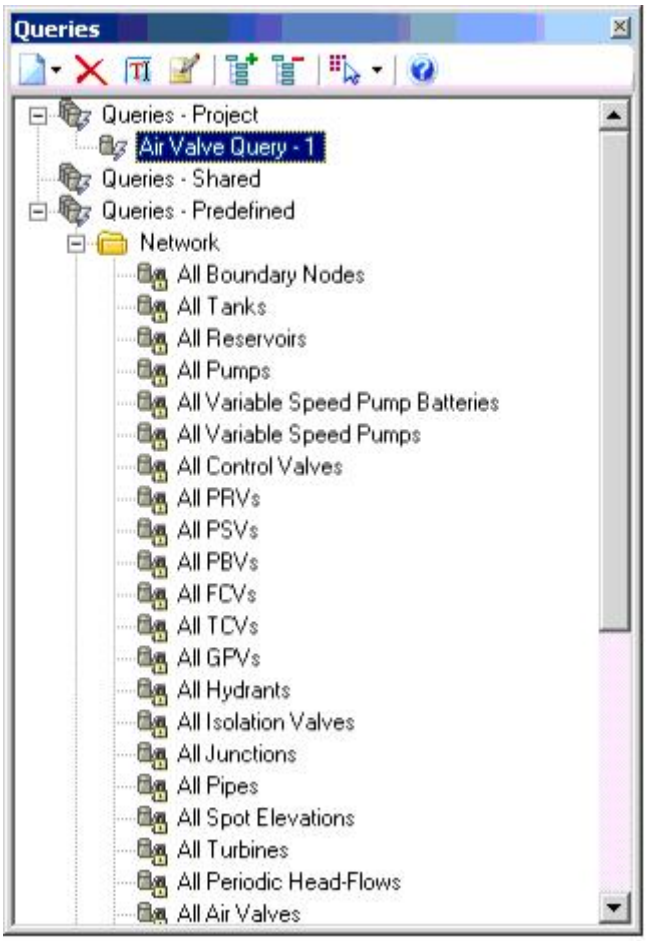
You can also use queries in the following ways:

- Create dynamic selection sets based on one or more queries. For more information, see “To create a Selection Set from a Query” on in the WaterGEMS V8i User’s Guide.
- Filter the data in a FlexTable using a query. For more information, see Sorting and Filtering FlexTable Data in the WaterGEMS V8i User’s Guide.
- You can use predefined queries in the Network Navigator. See Section 5.4.4 for more details.

For more information on how to construct queries, see Creating Queries in 7.1.2.

The Queries Manager
The Queries Manager is a docking manager that displays all queries in the current project, including predefined, shared, and project queries. You can create, edit, or delete shared and project queries from within the Queries Manager, as well as use it to select all elements in your model that are part of the selected query.

To open the Queries manager, click the View menu and select the Queries command, press <Ctrl+5>, or click the Queries button  on the View toolbar.



The Queries manager consists of a toolbar and a tree view, which displays all of the queries that are associated with the current project.

The toolbar contains the following icons:

New 	Contains the following commands: • **Query**—Creates a new SQL expression as either a project or shared query, depending on which item is highlighted in the tree view. • **Folder**—Creates a folder in the tree view, allowing you to group queries. You can right-click a folder and create queries or folders in that folder.
Delete 	Deletes the currently-highlighted query or folder from the tree view. When you delete a folder, you also delete all of the queries it contains.
Rename 	Renames the query or folder that is currently highlighted in the tree view.
Edit 	Opens the Query Builder dialog box, allowing you to edit the SQL expression that makes up the currently-highlighted query.

Expand All 	Opens all the Queries within all of the folders.
Collapse All 	Closes all the Query folders.
Select in Drawing 	Opens a submenu containing the following options: • **Select in Drawing** – Selects the element or elements that satisfy the currently highlighted query. • **Add to Current Selection** – Adds the element or elements that satisfy the currently highlighted query to the group of elements that are currently selected in the Drawing Pane. • **Remove from Current Selection** – Removes the element or elements that satisfy the currently highlighted query from the group of elements that are currently selected in the Drawing Pane. • **Select within Current Selection** – Selects the element or elements that both satisfy the current query and are already selected in the Drawing Pane.
Help 	Displays online help for the Query Manager.

7.1.2 Creating Queries

A query is a valid SQL expression that you construct in the Query Builder dialog box. You create and manage queries in the Query Manager. You also use queries to filter FlexTables and as the basis for a selection set.

To create a query from the Query Manager

1. Choose View > Queries or click the Queries icon on the View toolbar, or press <CTRL+5**>**.
2. Perform one of the following steps:
   - To create a new project query, highlight **Queries - Project** in the list pane, then click the **New** button and select **Query**.
   - To create a new shared query, highlight **Queries - Shared** in the list pane, then click the **New** button and select **Query**.

   Note: You can also right-click an existing item or folder in the list pane and select New > Query from the shortcut menu.

3. In the Select Element Type dialog box, select the desired element type from the drop-down menu. The Query Builder dialog box opens.
4. All input and results fields for the selected element type appear in the Fields list pane; available SQL operators and keywords are represented by buttons; and available

values for the selected field are listed in the Unique Values list pane. Perform the following steps to construct your query:

a. Double-click the field you wish to include in your query. The database column name of the selected field appears in the preview pane.
b. Click the desired operator or keyword button. The SQL operator or keyword is added to the SQL expression in the preview pane.
c. Click the **Refresh** button above the Unique Values list pane to see a list of unique values available for the selected field. Note that the Refresh button is disabled after you use it for a particular field (because the unique values do not change in a single query-building session).
d. Double-click the unique value you want to add to the query. The value is added to the SQL expression in the preview pane.

   Note: You can also manually edit the expression in the preview pane.

e. Click the **Validate** button above the preview pane to validate your SQL expression. If the expression is valid, the word "VALIDATED" is displayed in the lower right corner of the dialog box.
f. Click the **Apply** button above the preview pane to execute the query. If you didn't validate the expression, the Apply button validates it before executing it.
g. Click **OK**.

Perform these optional steps in the Query Manager:

- To create a new folder in the tree view, highlight the existing item or folder in which to place the new folder, then click the **New** button and select **Folder**. You can create queries and folders within folders.
- To delete an existing query or folder, click the **Delete** button. When you delete a folder, you also delete all of its contents (the queries it contains).
- To rename an existing query or folder, click the **Rename** button, and then type a new name.
- To edit the SQL expression in a query, select the query in the list pane, then click the **Edit** button. The Query Builder dialog box opens.
- To quickly select all the elements in the drawing pane that are part of the currently highlighted query, click the **Select in Drawing** button.

Example Query

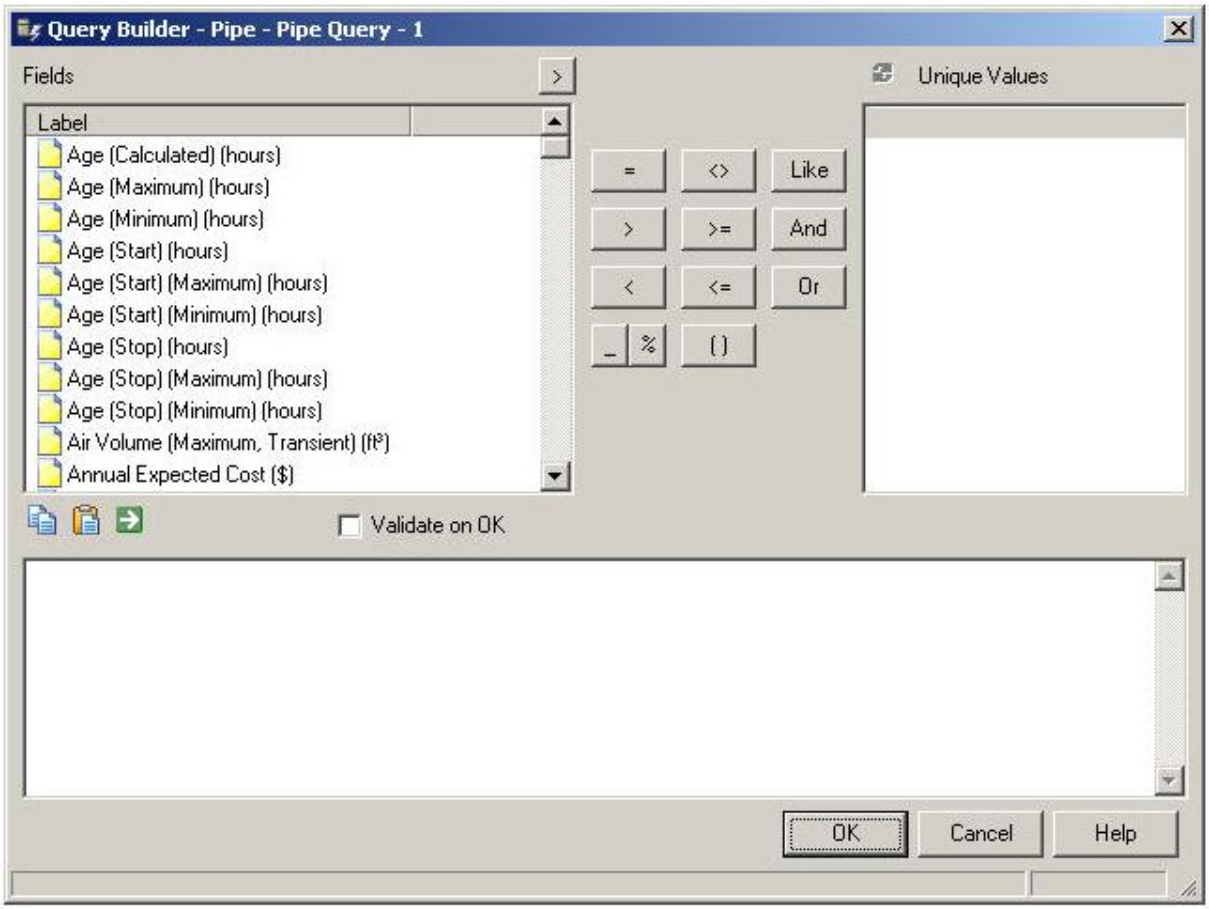
To create a query that finds all pipes with a diameter greater than 8 inches and less than or equal to 12 inches you would do the following:

1. In the **Queries** dialog, click the **New** button and select **Query**.
2. In the **Queries - Select Element Type** dialog, select **Pipe** and click **OK**.
3. In the **Query Builder** dialog, click the **()** (Parentheses) button.
4. Double-click **Diameter** in the **Fields** list.
5. Click the **>** (Greater Than) button.
6. Click the **Refresh** button above the **Unique Values** list. Double-click the value **8**.
7. In the Preview Pane, click to the right of the closing parenthesis.
8. Click the **And** button.
9. Click the **()** (Parentheses) button.
10. Double-click **Diameter** in the **Fields** list.
11. Click the **<=** (Less Than or Equal To) button.
12. Double-click the value **12** in the **Unique Values** list.

The final query will look like this:
(Physical_PipeDiameter > 8) AND (Physical_PipeDiameter <= 12)

Query Builder Dialog Box

You construct the SQL expression that makes up your query in the Query Builder dialog box. The Query Builder dialog box is accessible from the Query manager and from within a FlexTable.



The top part of the dialog box contains all the controls you need to construct your query: a list pane displaying all available attributes for the selected element type, an SQL control panel containing available SQL keywords and operators, and a list view that displays all the available values for the selected attribute. The bottom part of the dialog box contains a preview pane that displays your SQL expression as you construct it.

All the dialog box controls are described in the following table.

Fields 	Lists all input and results fields applicable to the selected element type. This list displays the labels of the fields while the underlying database column names of the fields become visible in the preview pane when you add them to the expression. Double-click a field to add it to your SQL expression.
SQL Controls	These buttons represent all the SQL operators and controls that you can use in your query. They include **=**, **>**, **<**, **_**, **%**, **<>**, **>=**, **<=**, **[ ]**, **Like**, **And**, and **Or**. Click the appropriate button to add the operator or keyword to the end of your SQL expression, which is displayed in the preview pane.

	
Unique Values	When you click the **Refresh** button, this list displays all the available unique values for the selected field. Double-click a value in the list to add it to the end of your SQL expression, which is displayed in the preview pane. If you select a different field, you must click the **Refresh** button again to update the list of unique values for the selected field. When you first open the Query Builder dialog box, this list is empty.
Refresh 	Updates the list of unique values for the selected field. This button is disabled after you use it for a particular field.
Copy 	Copies the entire SQL expression displayed in the preview pane to the Windows clipboard.
Paste 	Pastes the contents of the Windows clipboard into the preview pane at the location of the text cursor. For example, if your cursor is at the end of the SQL expression in the preview pane and you click the **Paste** button, the contents of your clipboard will be added to the end of the expression.
Validate on OK 	Turn on to validate the SQL expression in the preview pane. If the expression is not valid, a message appears. When you turn on and your SQL expression passes validation, the word "VALIDATED" appears in the lower right corner of the dialog box.
Apply 	Executes the query. The results of the query are displayed at the bottom of the Query Builder dialog box in the form "*x* of *x* elements returned."
Preview Pane	Displays the SQL expression as you add fields, operators and/or keywords, and values to it.
Action	Allows you to select the operation to be performed on the elements returned by the query defined in the Preview pane. The following choices are available: • **Create New Selection** – Creates a new selection containing the elements returned by the query. • **Add to Current Selection** – Adds the elements returned by the query to the current selection. • **Remove from Current Selection** – Removes the elements returned by the query from the current selection. • **Select Within Current Selection** – Selects the element or elements that both satisfy the current query and are already selected in the Drawing Pane. This control is only available when the Query Builder is accessed from the command Edit > Select By Attribute.

Note: If you receive a Query Syntax Error message notifying you that the query has too few parameters, check the field name you entered for typos. This message is triggered when the field name is not recognized.

8 WORKING WITH RESULTS

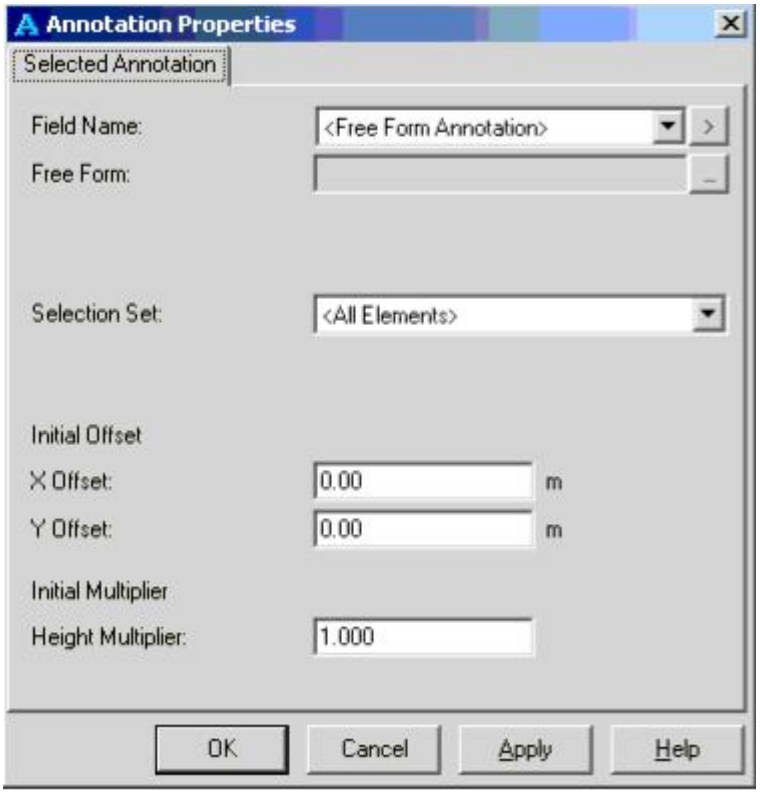
An important feature in all water distribution modeling software is the ability to present results clearly.

Bentley WaterGEMS V8*i* reporting features:

- **Reports**, which display and print information on any or all elements in the system.
- **Element Tables (FlexTables)**, for viewing, editing, and presentation of selected data and elements in a tabular format.
- **Profiles**, to graphically show, in a profile view, how a selected attribute, such as hydraulic grade, varies along an interconnected series of pipes.
- **Contouring**, to show how a selected attribute, such as pressure, varies throughout the distribution system.
- **Element Annotation**, for dynamic presentation of the values of user-selected variables in the plan view.
- **Color Coding**, which assigns colors based on ranges of values to elements in the plan view. Color coding is useful in performing quick diagnostics on the network.

When you want to label network attributes use the Annotation feature. With it, you can control which values are displayed, how they are labeled, and how units are expressed.

1. Choose View > Element Symbology > New > New Annotation



2. Select the Field Name to annotate.

3. Enter additional information into the other fields as needed.

• **Field Name**	Specify the attribute that is displayed by the annotation definition.
• **Free Form**	This field is only available when <Free Form Annotation> is selected in the Field Name list. Click the ellipsis button to open the Free Form Annotation dialog box.
• **Prefix**	Specify a prefix that is displayed before the attribute value annotation for each element to which the definition applies.
• **Suffix**	Specify a suffix that is displayed after the attribute value annotation for each element to which the definition applies. *Note: If you add an annotation that uses units, you can type "%u" in the prefix or suffix field to display the units in the drawing pane.*
• **Selection Set**	Specify a selection set to which the annotation settings will apply. If the annotation is to be applied to all elements, select the <All Elements> option in this field. <All Elements> is the default setting.
• **Initial Offset Checkbox**	When this box is checked, changes made to the X and Y Offset will be applied to current and subsequently created elements. When the box is unchecked, only subsequently created elements will be affected.
• **Initial X Offset**	Displays the initial X-axis offset of the annotation in feet. Sets the initial horizontal offset for an annotation. Set this at the time you create the annotation. Clicking OK will cause the new value to be used for all subsequent elements that you place. Clicking Apply will cause the new value to be applied to all elements.
• **Initial Y Offset**	Displays the initial Y-axis offset of the annotation in feet. Sets the initial vertical offset for an annotation. Set this at the time you create the annotation. Clicking OK will cause the new value to be used for all subsequent elements that you place. Clicking Apply will cause the new value to be applied to all elements.
• **Initial Multiplier Checkbox**	When this box is checked, changes made to the Height Multiplier will be applied to current and subsequently created elements. When the box is unchecked, only subsequently created elements will be affected.
• **Initial Height Multiplier**	Sets the initial size of the annotation text. Set this at the time you create the annotation. Clicking OK will cause the new value to be used for all subsequent elements that you place. Clicking Apply will cause the new value to be applied to all elements.

4. Click Apply.
5. The drawing will now display all of the annotations. You can try changing the properties of an element and recalculating. The annotations will update automatically to reflect any changes in the system.
6. If the annotation is crowded, you can click and drag the annotation to move it.
7. Click OK.

Drawing Style
Elements can be displayed in one of two styles in the Stand-Alone version; GIS style or CAD style.

Under GIS style, the size of element symbols in the drawing pane will remain the same (relative to the screen) regardless of zoom level. Under CAD style, element symbols will

Table Type	Specifies the type of elements that appear in the table. It also provides a filter for the attributes that appear in the Available Columns list. When you choose a table type, the available list only contains attributes that can be used for that table type. For example, only manhole attributes are available for a manhole table.

appear larger or smaller (relative to the drawing) depending on zoom level.

There is a default Drawing Style that is set on the Global tab of the Options dialog. The drawing style chosen there will be used by all elements by default. Changing the default drawing style will affect only new projects, not existing ones.

You can change the drawing style used by all of the elements in the project, or you can set each element individually to use either drawing style.

To change a single element's drawing style

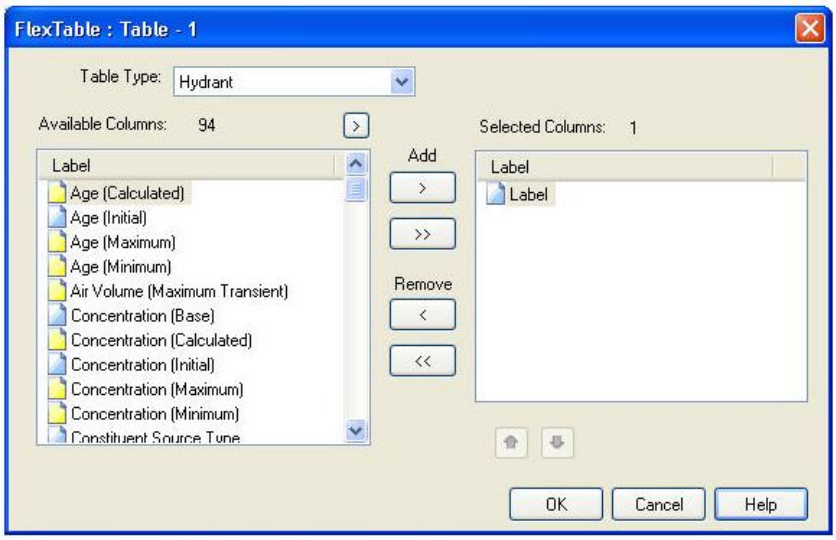
1. Double-click the element in the Element Symbology manager dialog to open the Properties manager.
2. In the Properties manager, change the value in the Display Style field to the desired setting.

To change the drawing style of all elements

Click the Drawing Style button in the Element Symbology manager and select the desired drawing style from the submenu that appears.

8.1 Tables

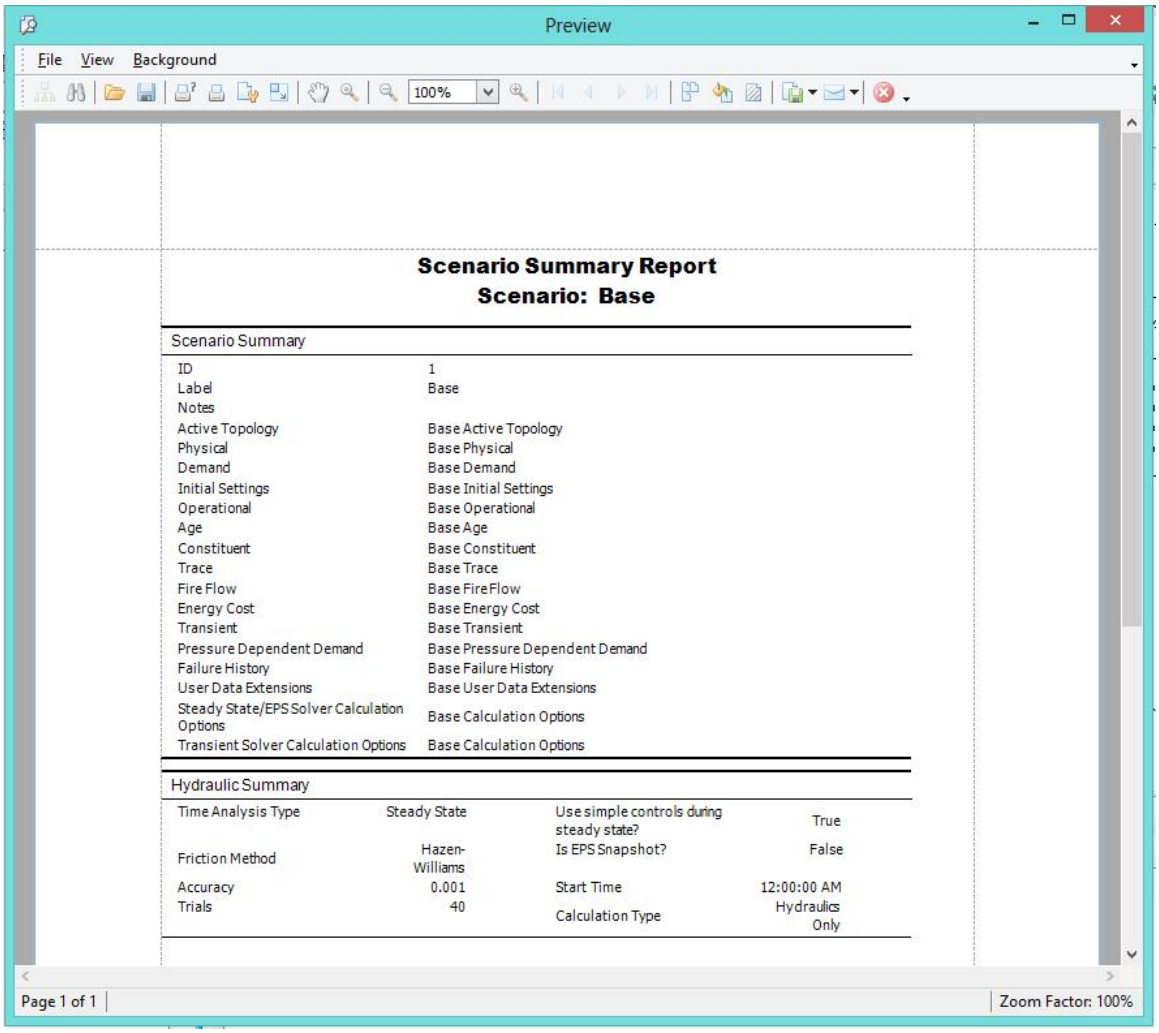
The Table Setup dialog box is where you can customize tables through the following options.



Available Columns	Contains all the attributes that are available for your table design. The Available Columns list is located on the left side of the Table Setup dialog box. This list contains all of the attributes that are available for the type of table you are creating. The attributes displayed in yellow represent non-editable attributes, while those displayed in white represent editable attributes. Click the Arrow button [>] to open a submenu that contains all of the available fields grouped categorically.
Selected Columns	Contains attributes that appear in your custom designed FlexTable. When you open the table, the selected attributes appear as columns in the table in the same order that they appear in the list. You can drag and drop or use the up and down buttons to change the order of the attributes in the table. The Selected Columns list is located on the right-hand side of the Table Setup dialog box. To add columns to the Selected Columns list, select one or more attributes in the Available Columns list, then click the Add button [>].
Add and Remove Buttons 	Select or clear columns to be used in the table and arrange the order the columns appear. The Add and Remove buttons are located in the center of the Table Setup dialog box. • [>] Adds the selected items from the Available Columns list to the Selected Columns list. • [>>] Adds all of the items in the Available Columns list to the Selected Columns list. • [<] Removes the selected items from the Selected Columns list. • [<<] Removes all items from the Selected Columns list. To rearrange the order of the attributes in the Selected Columns list, select the item to be moved, then click the up or down button. 

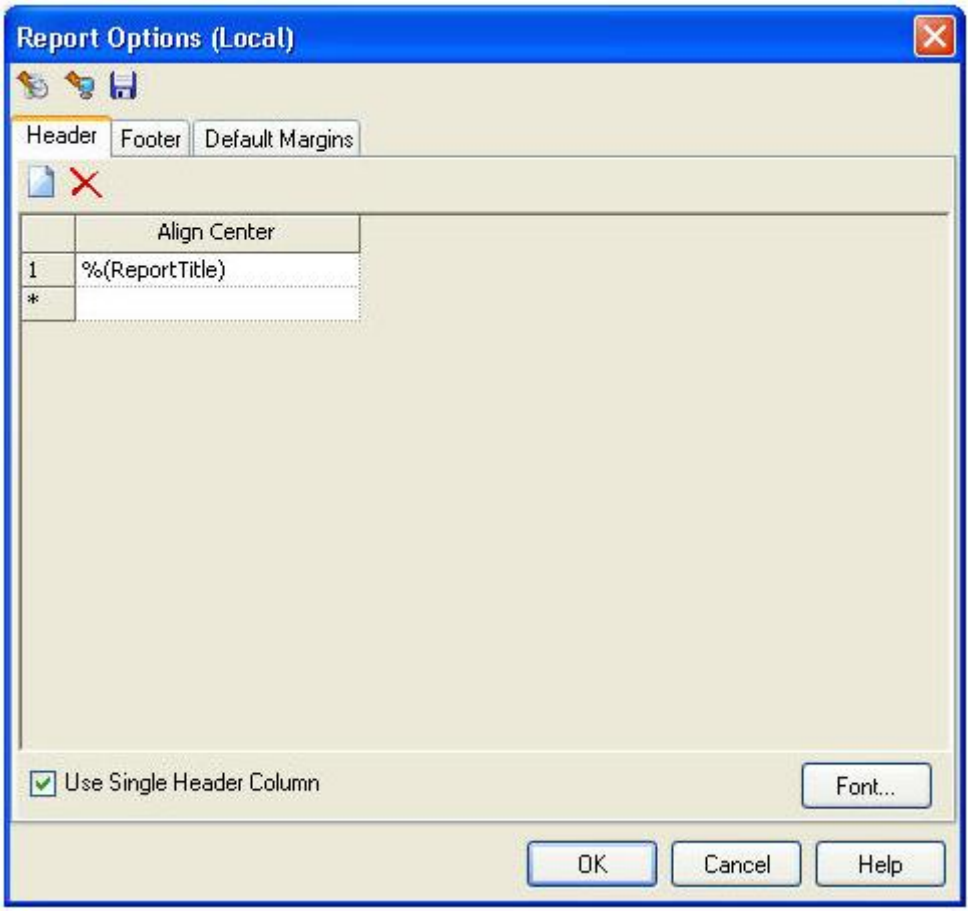
8.2 Reports

- Element Details
- Element Results
  - ✓ Select one or more elements
- Scenario Summary
- Project Inventory


Preview
File View Background
100%
Scenario Summary Report
Scenario: Base
Scenario Summary
ID 1
Label Base
Notes
Active Topology Base Active Topology
Physical Base Physical
Demand Base Demand
Initial Settings Base Initial Settings
Operational Base Operational
Age Base Age
Constituent Base Constituent
Trace Base Trace
Fire Flow Base Fire Flow
Energy Cost Base Energy Cost
Transient Base Transient
Pressure Dependent Demand Base Pressure Dependent Demand
Failure History Base Failure History
User Data Extensions Base User Data Extensions
Steady State/EPS Solver Calculation Options Base Calculation Options
Transient Solver Calculation Options Base Calculation Options
Hydraulic Summary
Time Analysis Type Steady State
Friction Method Hazen-Williams
Accuracy 0.001
Trials 40
Use simple controls during steady state? True
Is EPS Snapshot? False
Start Time 12:00:00 AM
Calculation Type Hydraulics Only
Page 1 of 1
Zoom Factor: 100%

Report Options

The Report Options dialog box offers control over how a report is displayed.



The header and footer can be fully customized and you can edit text to be displayed in the cells or select a pre-defined dynamic variable from the cell's menu.

- %(Company) - The name specified in the project properties.
- % (DateTime) - The current system date and time.
- % (BentleyInfo) - The standard Bentley company information.
- % (BentleyName) - The standard Bentley company name information.
- % (Pagination) - The report page out of the maximum pages.
- % (ProductInfo) - The current product and its build number.
- % (ProjDirectory) - The directory path where the project file is stored.
- % (ProjEngineer) - The engineer specified in the project properties.
- % (ProjFileName) - The full file path of the current project.
- % (ProjStoreFileName) - The full file path of the project.
- % (ProjTitle) - The name of the project specified in the project properties.
- % (ReportTitle) - The name of the report.
- %(Image) - Allows you to browse to and attach an image to the report header.
- % (AcademicLicense) - Adds text string: **Licensed for Academic Use Only**.
- % (HomeUseLicense) - Adds text string: **Licensed for Home Use Only**.
- % (ActiveScenarioLabel) - The label of the currently active scenario.

You can also select fonts, text sizes, and customize spacing, as well as change the default margins in the Default Margins tab.

8.3 Graphs

Use graphs to visualize your model or parts of your model, such as element properties or results. The model needs to be computed before you can create graphs. After you set up your elements and their properties, click the **Compute** button .

After the model has been calculated, you can graph elements directly from the drawing view.

To graph a single element Right-click an element in the drawing view and select the Graph command.

To graph a group of elements

1. Select a group of elements by drawing a selection box around them or by holding down the Ctrl key and then clicking a series of elements.
2. Right-click one of the selected elements and select the Graph command.

To Graph the elements contained in a selection set

1. Click the View menu and choose the Selection Sets command.
2. In the Selection Sets dialog, highlight the selection set to be graphed and click the Select in Drawing button.
3. Right-click one of the selected elements and select the Graph command.

8.3.1 Graph Manager

The Graph Manager contains any graphs you have created and saved in the current session or in a previous session. Graphs listed in the Graph Manager retain any customizations you have applied. You can graph computed values, such as flow and velocity.

To use the Graph Manager

1. Compute your model and resolve any errors.
2. Open the Graph Manager, click **View > Graphs**.
3. To Create a Graph select the elements that you want included from the drawing. Once you have selected the element you can either Right-click an element and select **Graph** or select the type of graph from the New button menu.

8.3.2 Printing a Graph

To print a graph click , or click **Print** **Preview** to view your graph then click print.

8.3.3 Working with Graph Data: Viewing and Copying

You can view the data that your graphs are based on. To view your data; create a graph; after the Graph dialog box opens, click the Data tab.

You can copy this data to the Windows clipboard for use in other applications, such as word-processing software.

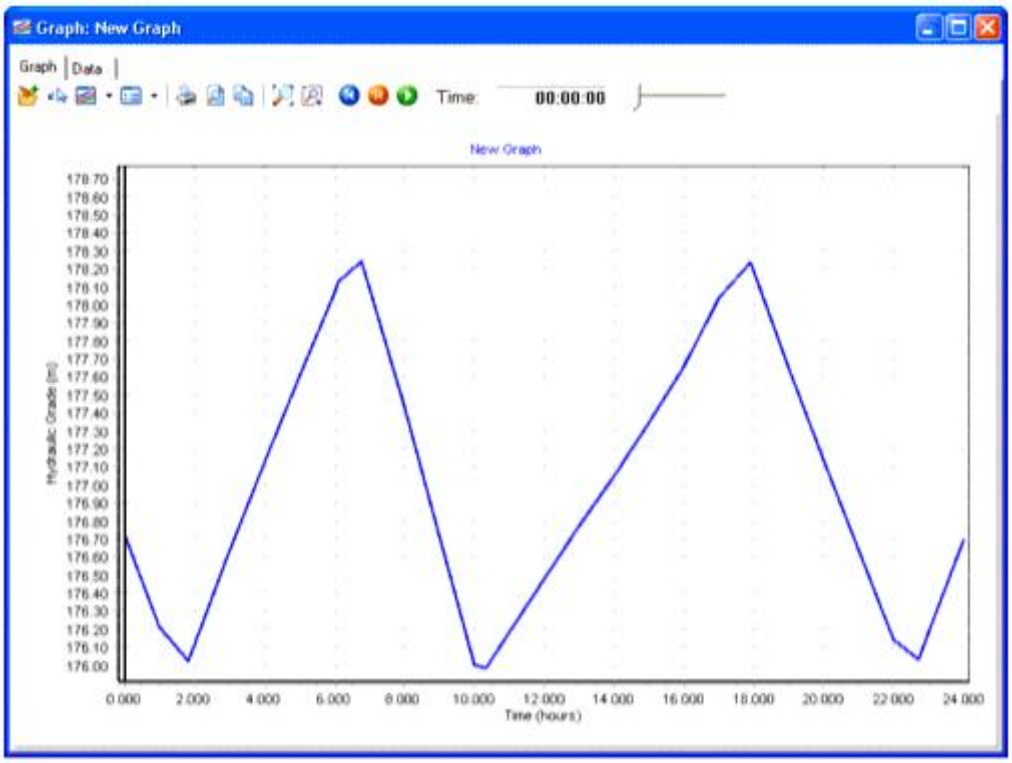
To copy this data

1. Click in the top-most cell of the left-most column to select the entire table, click a column heading to select an entire column, or click a row heading to select an entire row.
2. Press <**Ctrl+C>** to copy the selected data to the clipboard.
3. As needed, press <**Ctrl+V>** to paste the data as tab-delimited text into other software.

To print out the data for a graph, copy and paste it into another application, such as word-processing software or Notepad, and print the pasted content.

8.3.4 Graph Dialog Box

Using the Graph dialog box you can view and modify graph settings. After you create a graph, you view it in the Graph dialog box.

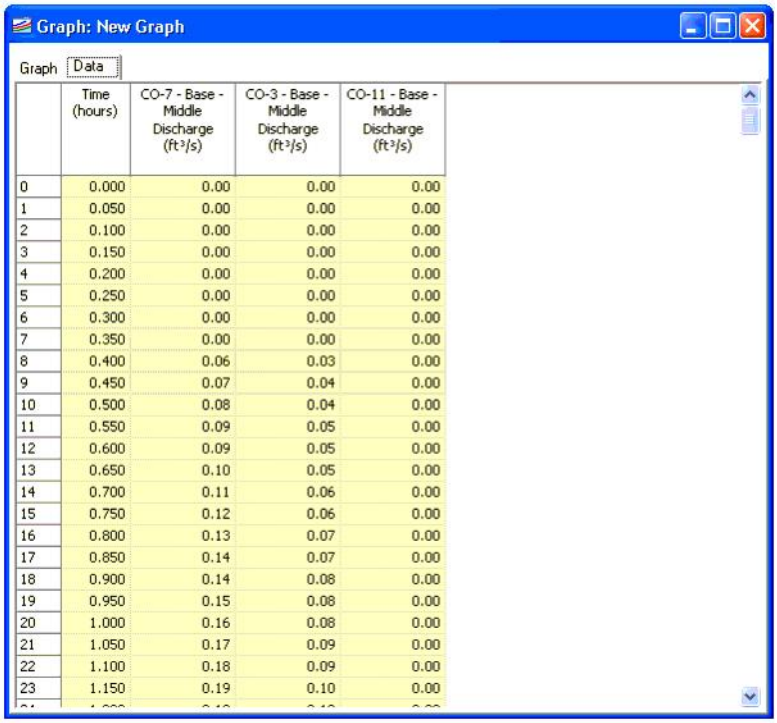


The following controls are available:

Add to Graph Manager 	Saves the Graph to the Graph Manager. When you click this button, the graph options (i.e., attributes to graph for a specific scenario) and the graph settings (i.e., line color, font size) are saved with the graph. If you want to view a different set of data (for example, a different scenario), you must change the scenario in the Graph Series Options dialog box. Graphs that you add to the Graph Manager are saved when you save your model, so that you can use the graph after you close and reopen Bentley WaterGEMS V8*i*.
Add to Graph 	Adds new elements to the graph using the current graph series options. Clicking this button returns you to the drawing view and opens a Select toolbar, allowing you to change which elements are included in the graph.

Graph Series Options 	Selects Graph Series Options to control what the graph displays. Select Observed Data to display user-defined attribute values alongside calculated results in the graph display dialog.
Chart Settings 	Opens a submenu containing the following commands: • Chart Options— Change graph display settings. • Detailed Labels—Click to view more information on the graph. • Legend-Click to view a legend for the graph. • Save Chart Options as Default—Saves the current chart options as the new default settings for future graphs. • Apply Default Chart Options—Applies the default chart options to the current graph. • Restore Factory Default Chart Options—Deletes the currently saved default chart options and replaces them with the default settings that were originally installed with WaterGEMS V8*i*.
Print 	Prints the current view in the graph display pane.
Print Preview 	Opens the Print Preview dialog box to view the current image and change the print information.
Copy 	Copies the current view in the graph display pane to the Windows Clipboard.
Zoom Extents 	Zooms out so that the entire graph is displayed.
Zoom 	Zooms in on a section of the graph. When the tool is toggled on, you can zoom in on any area of the graph by clicking on the chart to the left of the area to be zoomed, holding the mouse button, then dragging the mouse to the right (or the opposite extent of the area to be magnified) and releasing the mouse button when the area to be zoomed has been defined. To zoom back out, click and hold the mouse button, drag the mouse in the opposite direction (right to left), and release the mouse button.
Time (VCR) Controls 	Evaluate plots over time. • If you click Go to start, the Time resets to zero and the vertical line that marks time resets to the left edge of the Graph display. • If you click Pause, the vertical line that moves across the graph to mark time pauses, as does the Time field. • If you click Play, a vertical line moves across the graph and the

	Time field increments. The following controls are also available: • **Time**—Displays the time location of the vertical black bar in the graph display. This is a read-only field; to set a specific time, use the slider button. • **Slider**—Sets a specific time for the graph. A vertical line moves in the graph display and intersects your plots to show the value of the plot at a specific time. Use the slider to set a specific time value.
Graph Display Pane	Displays the graph.
Data Table	The Data tab displays the data that make up the graphs. If there is more than one item plotted, the data for each plot is provided. You can copy and paste the data from this tab to the clipboard for use in other applications, such as Microsoft Excel. To select an entire column or row, click the column or row heading. To select the entire contents of the Data tab, click the heading cell in the top-left corner of the tab. Use <Ctrl+C> and <Ctrl+V> to paste your data. The column and row headings are not copied. The Data tab is shown below.

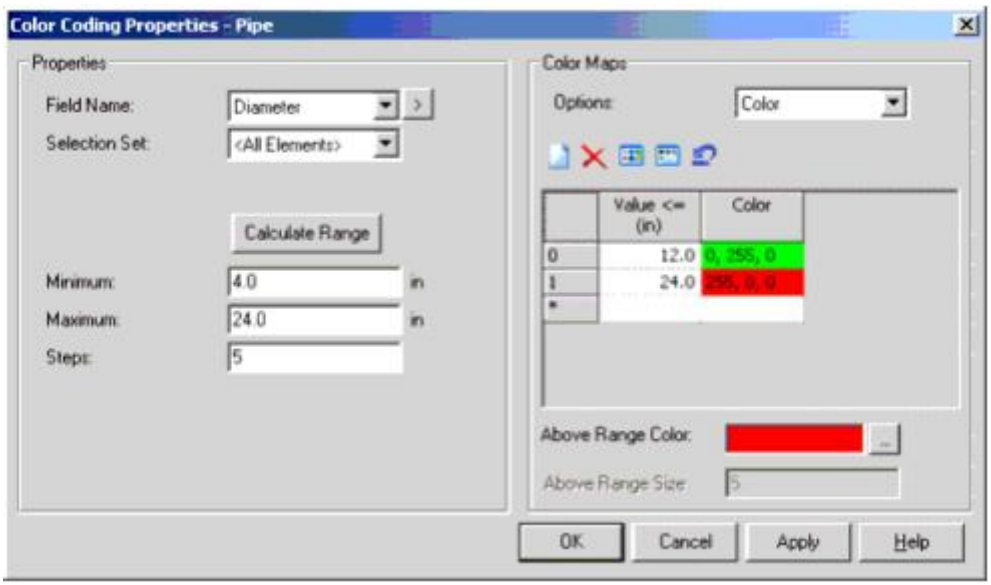


	Time (hours)	CO-7 - Base - Middle Discharge (ft³/s)	CO-3 - Base - Middle Discharge (ft³/s)	CO-11 - Base - Middle Discharge (ft³/s)
0	0.000	0.00	0.00	0.00
1	0.050	0.00	0.00	0.00
2	0.100	0.00	0.00	0.00
3	0.150	0.00	0.00	0.00
4	0.200	0.00	0.00	0.00
5	0.250	0.00	0.00	0.00
6	0.300	0.00	0.00	0.00
7	0.350	0.00	0.00	0.00
8	0.400	0.06	0.03	0.00
9	0.450	0.07	0.04	0.00
10	0.500	0.08	0.04	0.00
11	0.550	0.09	0.05	0.00
12	0.600	0.09	0.05	0.00
13	0.650	0.10	0.05	0.00
14	0.700	0.11	0.06	0.00
15	0.750	0.12	0.06	0.00
16	0.800	0.13	0.07	0.00
17	0.850	0.14	0.07	0.00
18	0.900	0.14	0.08	0.00
19	0.950	0.15	0.08	0.00
20	1.000	0.16	0.08	0.00
21	1.050	0.17	0.09	0.00
22	1.100	0.18	0.09	0.00
23	1.150	0.19	0.10	0.00

8.4 Color Coding

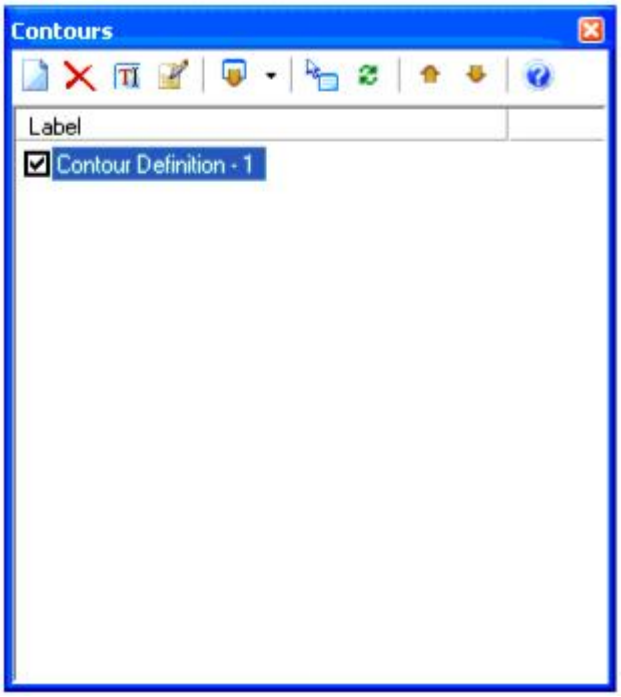
Use color coding to help you quickly see what's going on in your model or to change the color and/or size of elements based on the value of data that you select, such as flow or element size.

To work with color coding, go to View > Element Symbology > New Color Coding to open the Color Coding Properties dialog box.

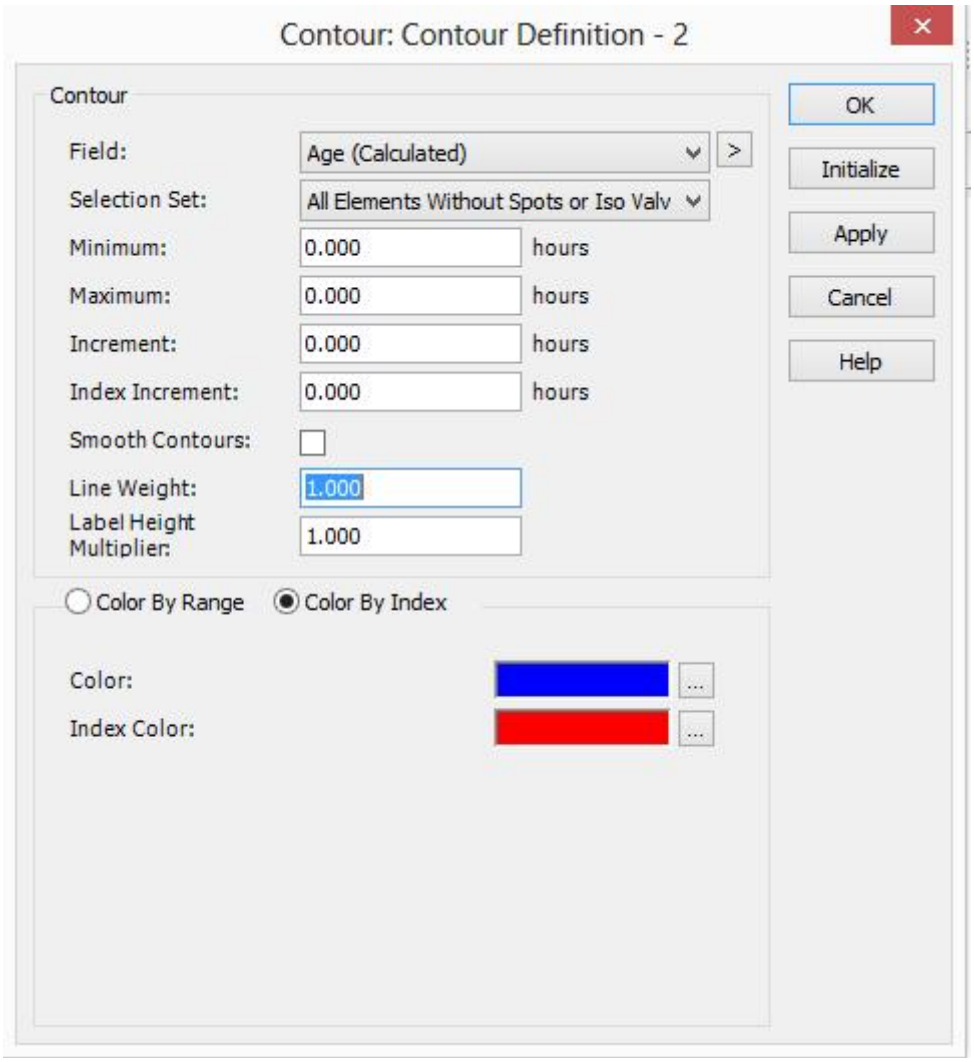


8.5 Contours

Using WaterGEMS V8i you can visually display calculated results for many attributes using contour plots.



The Contours dialog box is where all of the contour definitions associated with a project are stored. Choose View > Contours to open the Contours dialog box.



Field	Select the attribute to apply the contour.
Selection Set	Apply an attribute to a previously defined selection set or to one of the following predefined options: • **All Elements** - Calculates the contour based on all elements in the model, including spot elevations. • **All Elements Without Spots** - Calculates the contour based on all elements in the model, except for spot elevations.
Minimum	Lowest value to be included in the contour map. It may be desirable to use a minimum that is above the absolute minimum value in the system to avoid creating excessive lines near a pump or other high-differential portions of the system.
Maximum	Highest value for which contours will be generated.
Increment	Step by which the contours increase. The contours created will be evenly divisible by the increment and are not directly related to the minimum and maximum values. For example, a contour set with 10 minimum, 20 maximum, and an increment of 3 would result in the following set: [12, 15, 18] not [10, 13, 16, 19].
Index	Value for which contours will be highlighted and labeled. The index increment

Increment	should be an even multiple of the standard increment.
Smooth Contours	The Contour Smoothing option displays the results of a contour map specification as smooth, curved contours.
Line Weight	The thickness of contour lines in the drawing view.
Label Height Multiplier	When contours are created, there are labels (text) placed on the end of the index contours. This text has a default size. The Label Height Multiplier field allows you to scale the text size for these labels up/down.
Color by Range	Contours are colored based on attribute ranges. Use the Initialize button to create five evenly spaced ranges and associated colors.
Initialize	This button, located to the right of the Contour section, will initialize the Minimum, Maximum, Increment, and Index Increment values based on the actual values observed for the elements in the selection set. *Note: Initialization can be accomplished by clicking the Initialize button to automatically generate values for the minimum, maximum, increment, and index increment to create an evenly spaced contour set.*
Color by Index	The standard contours and index contours have separately controlled colors that you can make the contours more apparent.

Note: Setting up contours should be handled with care or useless information will be created.

9 ALTERNATIVES AND SCENARIOS

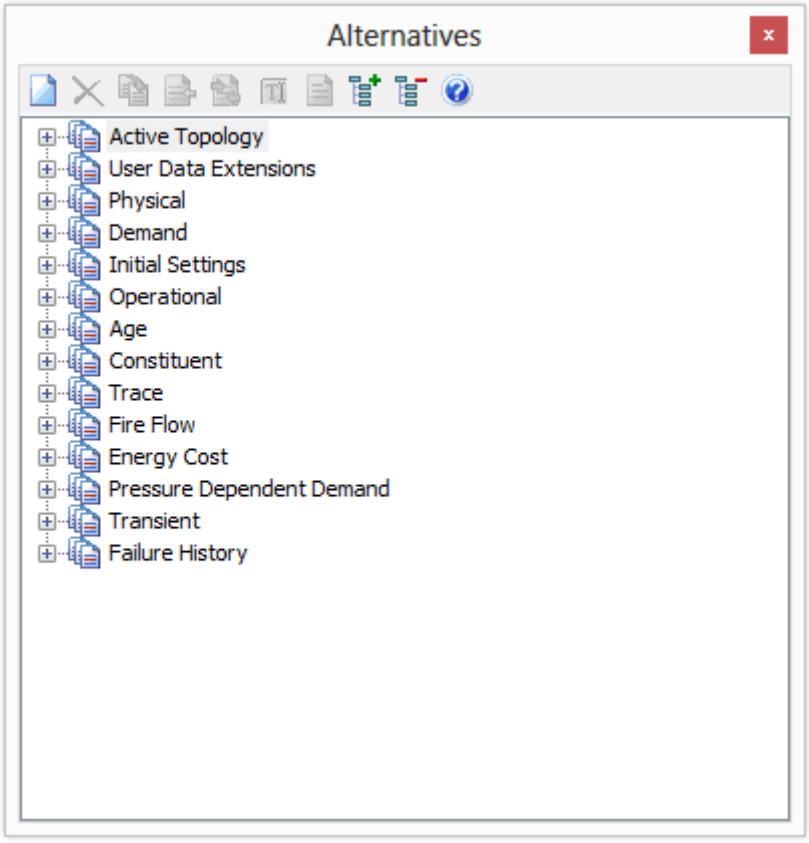
Alternatives are the building blocks behind scenarios. They are categorized data sets that create scenarios when placed together. Alternatives hold the input data in the form of records. A record holds the data for a particular element in your system.

Scenarios are composed of alternatives as well as other calculation options, allowing you to compute and compare the results of various changes to your system. Alternatives can vary independently within scenarios and can be shared between scenarios.

Scenarios allow you to specify the alternatives you want to analyze. In combination with scenarios, you can perform calculations on your system to see the effect of each alternative. Once you have determined an alternative that works best for your system, you can permanently merge changes from the preferred alternative to the base alternative.

When you first set up your system, the data that you enter is stored in the various base alternative types. If you want to see how your system behaves, for example, by increasing the diameter of a few select pipes, you can create a child alternative. You can make another child alternative with even larger diameters and another with smaller diameters. The number of alternatives that can be created is unlimited.

The Alternative Manager allows you to create, view, and edit the alternatives that make up the project scenarios. The dialog box consists of a pane that displays folders for each of the alternative types which can be expanded to display all of the alternatives for that type and a toolbar.



There are two kinds of alternatives: Base alternatives and Child alternatives. Base alternatives contain local data for all elements in your system. Child alternatives inherit data from base alternatives, or even other child alternatives, and contain data for one or more elements in your system. The data within an alternative consists of data inherited from its parent and the data altered specifically by you (local data).

Remember that all data inherited from the base alternative are changed when the base alternative changes. Only local data specific to a child alternative remain unchanged.

New alternatives are created in the Alternative Manager dialog box. A new alternative can be a Base scenario or a Child scenario. Each alternative type contains a Base alternative in the Alternative Manager tree view.

To create a new Alternative

1. Select **Analysis > Alternatives** to open the Alternative Manager, or click  To create a new Base alternative, select the type of alternative you want to create, then click the **New** button.
2. To create a new Child alternative, right-click the Base alternative from which the child will be derived, then select **New > Child Alternative** from the menu.
3. Double-click the new alternative to edit its properties.
4. Click Close when finished.

Types of Alternatives and how they are used

Active Topology. The Active Topology Alternative allows you to temporarily remove areas of the network from the current analysis. This is useful for comparing the effect of proposed construction and to gauge the effectiveness of redundancy that may be present in the system.

Physical Alternative
One of the most common uses of a water distribution model is the design of new or replacement facilities. During design, it is common to try several physical alternatives in an effort to find the most cost effective solution. For example, when designing a replacement pipeline, it would be beneficial to try several sizes and pipe materials to find the most satisfactory combination.

Demand Alternatives
The demand alternative allows you to model the response of the pipe network to different sets of demands, such as the current demand and the demand of your system ten years from now.

Initial Settings Alternative
The Initial Settings Alternative contains the data that set the conditions of certain types of network elements at the beginning of the simulation. For example, a pipe can start in an open or closed position and a pump can start in an on or off condition.

Operational Alternatives
The Operational Alternative is where you can specify controls on pressure pipes, pumps, as well as valves

Age Alternatives

The Age Alternative is used when performing a water quality analysis for modeling the age of the water through the pipe network. This alternative allows you to analyze different scenarios for varying water ages at the network nodes.

Constituent Alternatives
The Constituent Alternative contains the water quality data used to model a constituent concentration throughout the network when performing a water quality analysis

Trace Alternative
The Trace Alternative is used when performing a water quality analysis to determine the percentage of water at each node coming from a specified node. The Trace Alternative data includes a Trace Node, which is the node from which all tracing is computed.

Fire Flow Alternative
The Fire Flow Alternative contains the input data required to perform a fire flow analysis. This data includes the set of junction nodes for which fire flow results are needed, the set of default values for all junctions included in the fire flow set, and a record for each junction node in the fire flow set.

Energy Cost Alternative
The Energy Cost Alternative allows you to specify which tanks, pumps, and variable speed pump batteries will be included in the Energy Cost calculations. For pumps, you can also select which energy pricing pattern will be used or create a new one. You can also run a report.

Pressure Dependent Demand Alternative
The Pressure Dependent Demand Alternative allows a pressure dependent demand function to be used.

Transient Alternative
The Transient Alternative allows you to edit and view data that is used for WaterGEMS V8*i* transient calculations. There is a tab for each element type, each containing the WaterGEMS V8*i* specific attributes for that element type.

Failure History Alternative
The Failure History alternative allows you to edit data associated with pipe break analysis.

User Data Extensions
The User Data Alternative allows you to edit the data defined in the User Data Extension command for each of the network element types. The User Data Alternative editor contains a tab for each type of network element and is project specific.

9.1 Editing Alternatives

You edit the properties of an alternative in its own alternative editor. The first column in an alternative editor contains check boxes, which indicate the records that have been changed in this alternative.

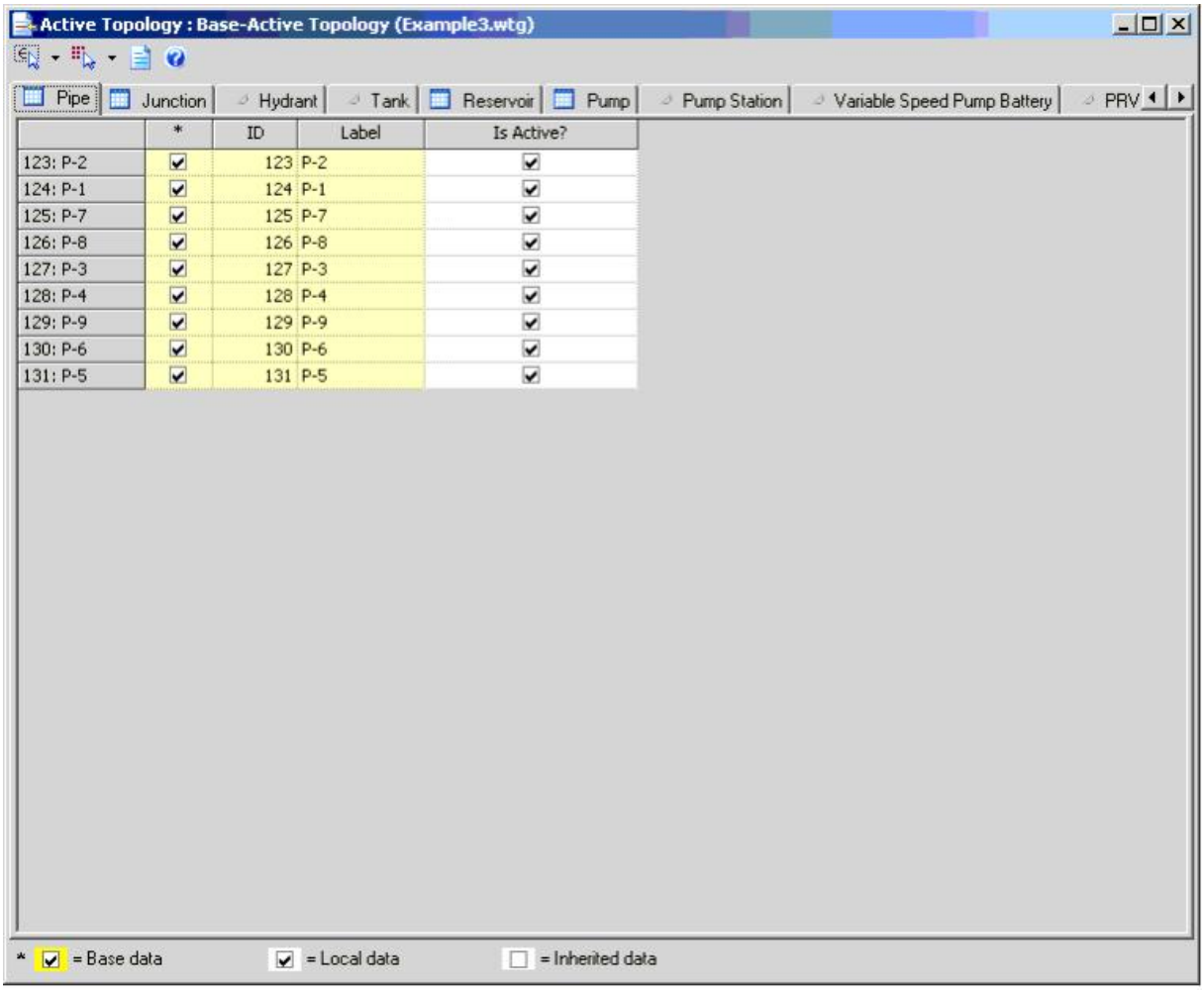
- If the box is checked, the record on that line has been modified and the data is local, or specific, to this alternative.
- If the box is not checked, it means that the record on that line is inherited from its higher-level parent alternative. Inherited records are dynamic. If the record is changed in the parent, the change is reflected in the child. The records on these rows reflect the corresponding values in the alternative's parent.

To edit an existing alternative, you can use one of two methods:

- Double-click the alternative to be edited in the Alternative Manager or
- Select the alternative to be edited in the Alternative Manager and click Edit 

Alternative Editor Dialog Box

This dialog box presents in tabular format the data that makes up the alternative being edited. Depending on the alternative type, the dialog box contains a separate tab for each element that possesses data contained in the alternative



The Alternative Editor displays all of the records held by a single alternative. These records contain the values that are active when a scenario referencing this alternative is active. They allow you to view all of the changes that you have made for a single alternative. They also allow you to eliminate changes that you no longer need.

There is one editor for each alternative type. Each type of editor works similarly and allows you to make changes to a different aspect of your system. The first column contains check boxes, which indicate the records that have been changed in this alternative.

If the check box is selected, the record on that line has been modified and the data is local, or specific, to this alternative.

If the check box is cleared, it means that the record on that line is inherited from its higher-level parent alternative. Inherited records are dynamic. If the record is changed in the parent, the change is reflected in the child. The records on these rows reflect the corresponding values in the alternative's parent.

When the editor has tabs for various element types, you can determine whether the alternative contains data for that element type by the icon next to the element type; if it is highlighted  , the alternative contains data for that element type. If the element type is not used in the current model the tab is marked with an icon .

Note: *As you make changes to records, the check box automatically becomes checked. If you want to reset a record to its parent's values, clear the corresponding check box.*

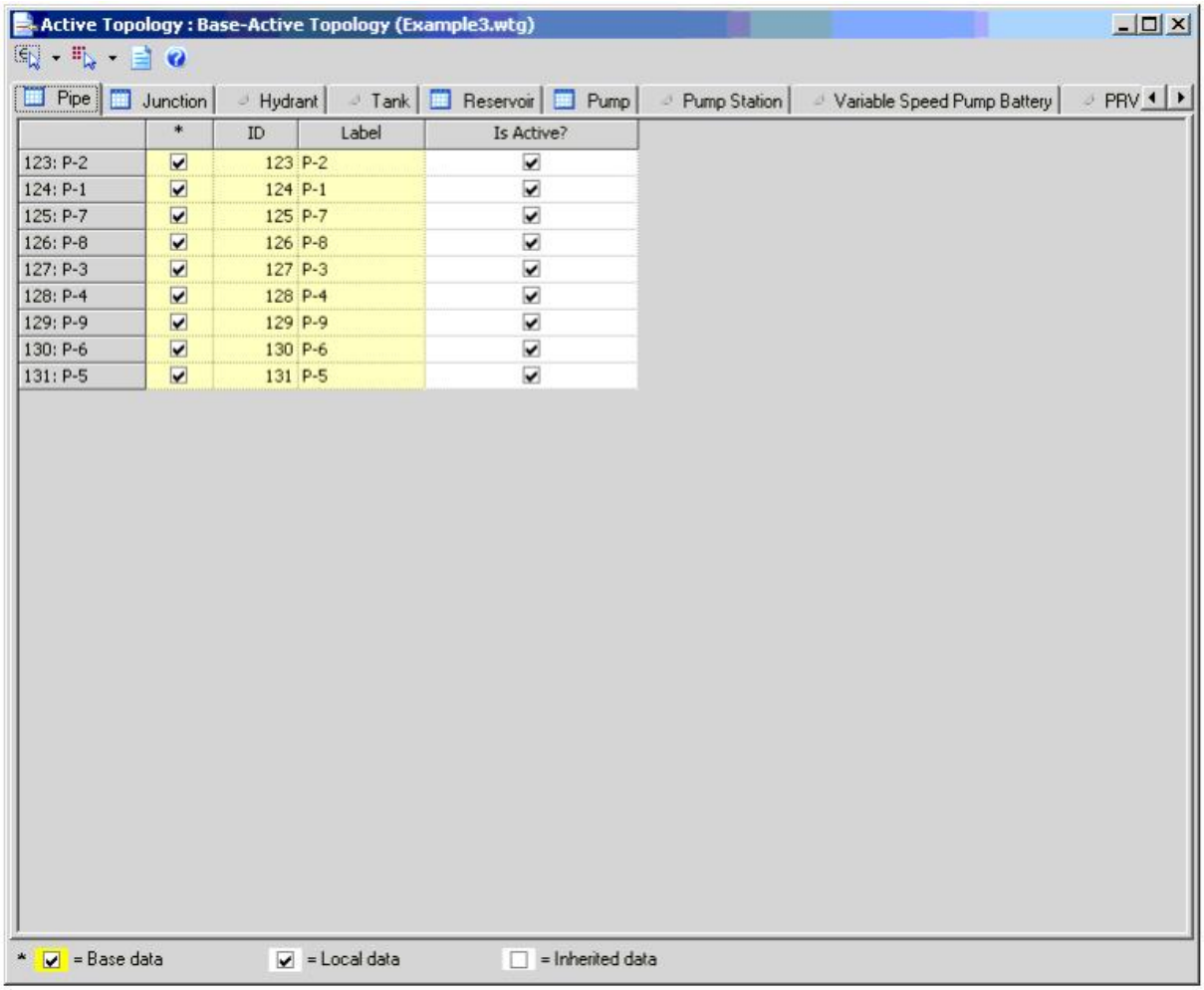
Many columns support Global Editing (see Globally Editing Data), allowing you to change all values in a single column. Right-click a column header to access the Global Edit option.

The check box column is disabled when you edit a base alternative.

In either case, the Alternative Editor dialog box for the specified alternative opens, allowing you to view and define settings as desired.

9.2 Active Topology Alternative

The Active Topology Alternative allows you to temporarily remove areas of the network from the current analysis. This is useful for comparing the effect of proposed construction and to gauge the effectiveness of redundancy that may be present in the system.



	*	ID	Label	Is Active?
123: P-2	☑	123	P-2	☑
124: P-1	☑	124	P-1	☑
125: P-7	☑	125	P-7	☑
126: P-8	☑	126	P-8	☑
127: P-3	☑	127	P-3	☑
128: P-4	☑	128	P-4	☑
129: P-9	☑	129	P-9	☑
130: P-6	☑	130	P-6	☑
131: P-5	☑	131	P-5	☑

For each tab, the same setup applies—the tables are divided into four columns. The first column displays whether the data is Base or Inherited, the second column is the element ID, the third column is the element Label, and the fourth column allows you to choose whether or not the corresponding element is Active in the current alternative.

To make an element Inactive in the current alternative, clear the check box in the **Is Active?** column that corresponds to that element's Label.

The following buttons are available:

Selection Set

Opens a submenu containing the following options:

- **Create Selection Set**—Allows you to create a new selection set.
- **Add to Selection Set**—Adds all of the elements in the current tab of the alternative to a previously created selection set that you specify.
- **Remove from Selection Set**——Removes all of the elements in the current tab of the alternative from a previously created selection set that you specify.

Select in Drawing

Opens a submenu containing the following options:

- **Select in Drawing**—Selects the elements in the current tab of the

alternative in the drawing pane.

- **Add to Current Selection**—Adds all of the elements in the current tab of the alternative to the group of elements that are currently selected in the Drawing Pane.
- **Remove from Current Selection**—Removes the elements in the current tab of the alternative from the group of elements that are currently selected in the Drawing Pane.

- **Select Within Current Selection**—Selects the element or elements that are both in the current tab of the alternative and are already selected in the Drawing Pane.

Report Generates a report containing the data within the current alternative.

Help Opens the online help.

Creating an Active Topology Child Alternative

When creating an active topology child alternative, you may notice that the elements added to the child scenario become available in your model when the base scenario is the current scenario.

To create an active topology alternative so that the elements added to the child scenario do not show up as part of the base scenario

1. Create a new WaterGEMS V8*i* project.
2. Open the Property Editor.
3. Open the Scenario Manager and make sure the Base scenario is current (active).
4. Create your model by adding elements in the drawing pane.
5. Create a new child scenario and a new child active topology alternative:
    a. In the Scenario Manager, click the **New** button and select **Child Scenario** from the submenu.
    b. The new Child Scenario is created and can be renamed.
    c. In the Alternatives Manager, open **Active Topology,** select the **Base Active Topology**, right-click to select **New**, then **Child Alternative**.
    d. Rename the new Child Alternative.
6. In the Scenario Manager, select the new child scenario then click **Make Current** to make the child scenario the current (active) scenario.
7. Add new elements to your model. These elements will be active only in the new child alternative.
8. To verify that this worked:
    a. In the Scenario Manager, select the base scenario then click **Make Current** to make the base scenario the current (active) scenario. The new elements are shown as inactive (they are grayed out in the drawing pane).

b. In the Scenario Manager, select the new child scenario then click **Make Current** to make the child scenario the current (active) scenario. The new elements are shown as active.

Note: If you add new elements in the base scenario, they will show up in the child scenario.

9.3 Physical Alternative

One of the most common uses of a water distribution model is the design of new or replacement facilities. During design, it is common to try several physical alternatives in an effort to find the most cost effective solution. For example, when designing a replacement pipeline, it would be beneficial to try several sizes and pipe materials to find the most satisfactory combination.

Each type of network element has a specific set of physical properties that are stored in a physical properties alternative. To access the Physical Properties Alternative select **Analysis > Alternatives** and select Physical Alternative.

9.4 Demand Alternatives

The demand alternative allows you to model the response of the pipe network to different sets of demands, such as the current demand and the demand of your system ten years from now.

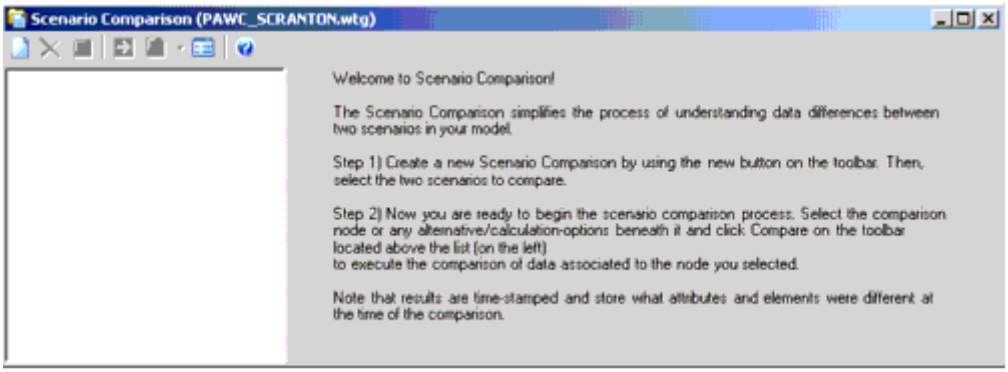
9.5 Scenario Comparison

The scenario comparison tool enables you to compare input values between any two scenarios to identify differences quickly. While WaterGEMS/CAD/HAMMER users have previously had the capability to open a child scenario or alternative and compare it with its parent, this tool greatly extends that capability in that you can compare any two scenarios or alternatives (not necessarily parent-child) and very easily detect differences.

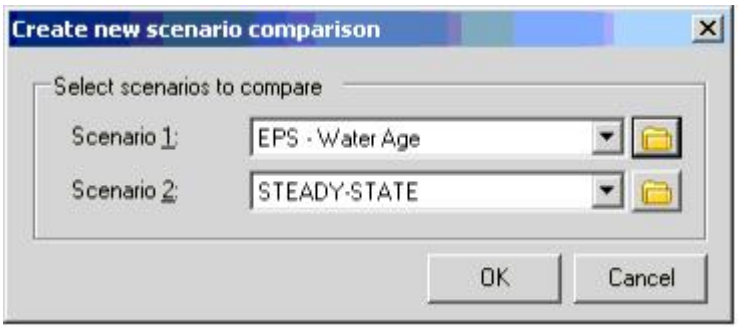
The scenario comparison tool can be started by picking Tools > Scenario Comparison or by

selecting the Scenario Comparison button from the toolbar . If the button is not visible, it can be added using the "Add or Remove Buttons" drop down from the Tools toolbar (see Customizing WaterGEMS V8i Toolbars and Buttons).

On first opening the scenario comparison tool, the dialog below opens which gives an overview of the steps involved in using the tool. Pick the New button (leftmost).

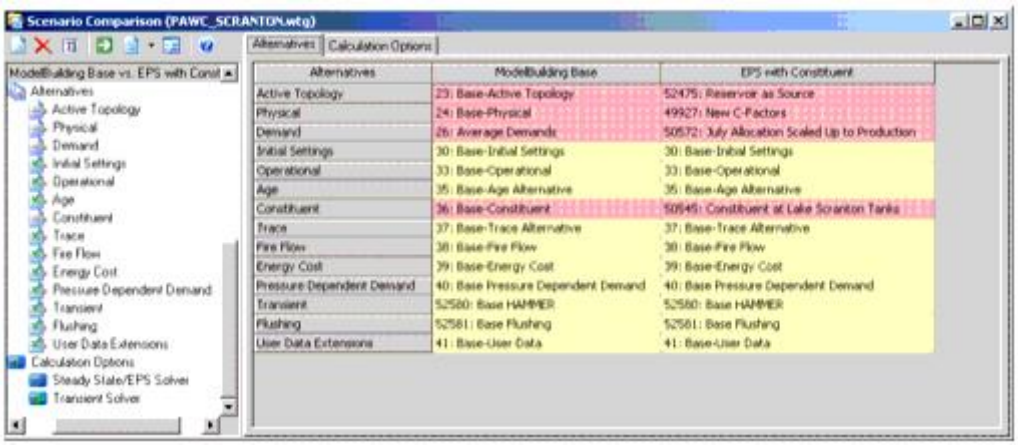


This opens a dialog which allows you to select which two scenarios will be compared.



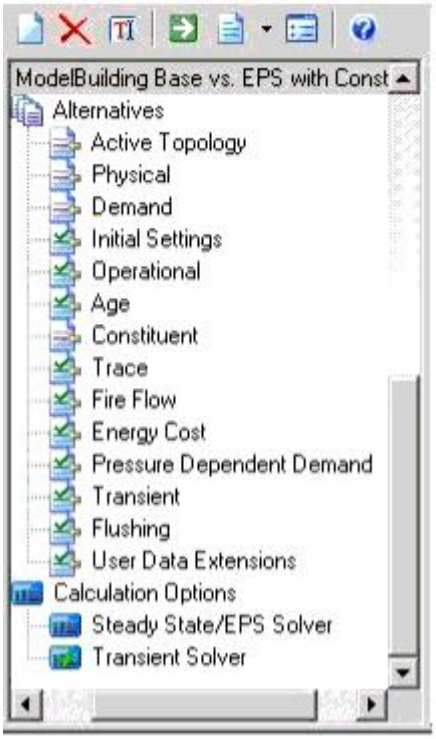
The scenario manager button next to each selection gives you the ability to see the tree view of scenarios. Choose OK to begin the scenario comparison tool. This initially displays a list of alternatives and calculation options, with the ones with identical properties displayed with a yellow background and those with different properties displayed with a pink background. The background color can be changed from pink to any other color by selecting the sixth button from the left and then selecting the desired color.

The dialog below shows that the Active Topology, Physical, Demand and Constituent alternatives are different between the scenarios. There is a second tab for Calculation Options which shows if the calculation options are different between scenarios.



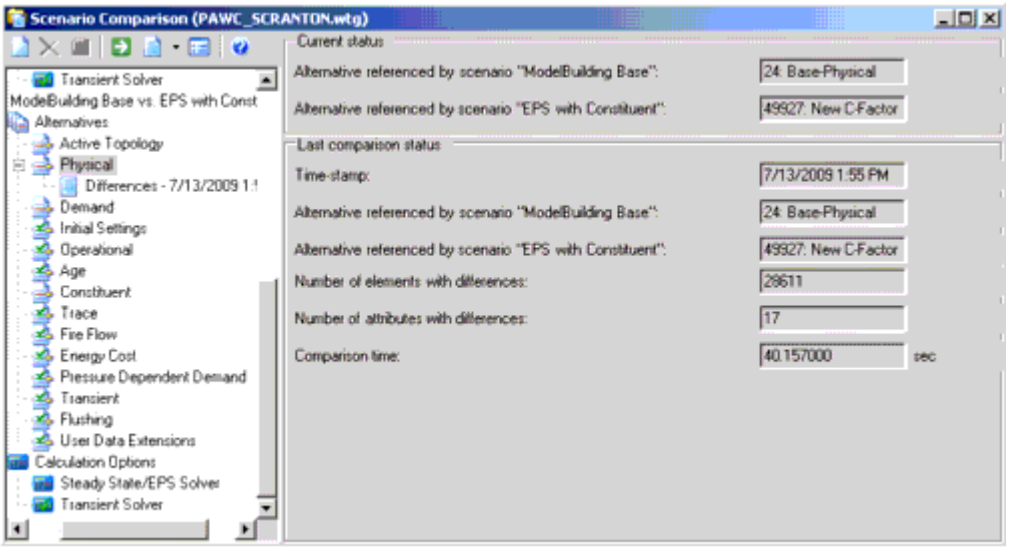
This display can also be copied to the clipboard using the Copy button.

The alternatives that have differences are also shown in the left pane with a red mark as opposed to the green check indicating that there are no differences.



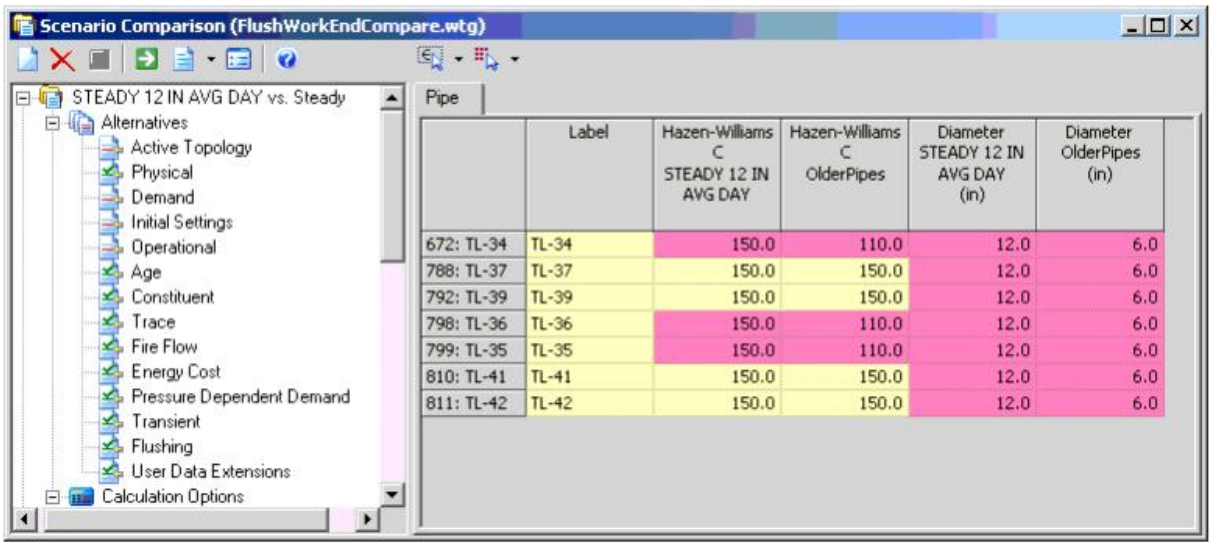
To obtain more detailed information on differences, highlight one of the alternatives and select the green and white Compute arrow at the top of pane (fourth button).

This initially returns a summary of the comparison which indicates the time when the comparison was run, which scenarios were involved and number of elements and attributes for which there were differences.



By picking "Differences" in the left pane for the alternative of interest, you can view the differences. In this display, only the elements and properties that are different are shown with

a pink background. In the example below, only 7 pipes had their diameters changed and only 3 of those had difference C-factors. There are separate tables for each element type that had differences.



	Label	Hazen-Williams C STEADY 12 IN AVG DAY	Hazen-Williams C OlderPipes	Diameter STEADY 12 IN AVG DAY (in)	Diameter OlderPipes (in)
672: TL-34	TL-34	150.0	110.0	12.0	6.0
788: TL-37	TL-37	150.0	150.0	12.0	6.0
792: TL-39	TL-39	150.0	150.0	12.0	6.0
798: TL-36	TL-36	150.0	110.0	12.0	6.0
799: TL-35	TL-35	150.0	110.0	12.0	6.0
810: TL-41	TL-41	150.0	150.0	12.0	6.0
811: TL-42	TL-42	150.0	150.0	12.0	6.0

Using the buttons on top of the right pane, when Differences is selected, you can create a selection set of the elements with differences or highlight those elements in the drawing. This is very useful for finding elements with differences in a large model.

10 RUNNING THE MODEL

The following steps need to be completed before performing hydraulic calculations for a network.

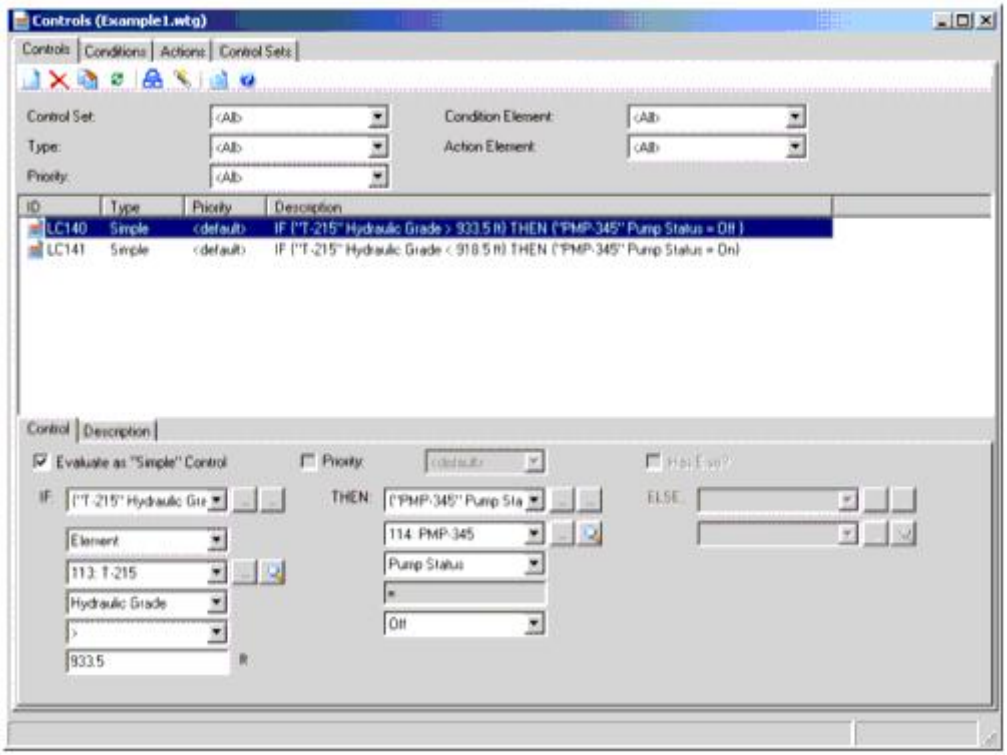
1. Click the Analysis toolbar and select Calculation Options.
2. In the Calculation Options dialog, double-click Base Calculation Options or create a new one and double-click it. This will open the Properties viewer.
3. In the Properties viewer, set the Time Analysis Type to Steady-State or Extended Period. If Extended Period is selected, then specify the starting time, the duration, and the time step to be used.
4. Optionally, in Extended Period mode, you may perform a Water Quality Analysis. Set the Calculation Type to Age, Constituent or Trace.
5. Optionally, in Steady-State mode, you may also perform a Fire Flow Analysis. Change the Calculation Type to Fire Flow.
6. Optionally, in the Adjustments section, you may modify the demand, unit demand, or roughness values of your entire network for calibration purposes. If Demand Adjustments, Unit Demand Adjustments, or Roughness Adjustments are set to Active in the Calculation Option properties and adjustments have been specified, the active adjustments will be used. This does not permanently change the value of the input data, but allows you to experiment with different calibration factors until you find the one that causes your calculation results to most closely correspond with your observed field data.
7. Optionally, verify and/or adjust the settings in Hydraulics section to change the general algorithm parameters used to perform Hydraulic and Water Quality calculations.
8. Click Validate  to ensure that your input data does not contain errors.
9. Click Compute  to start the calculations.

10.1 Controls

Controls give you a way to specify for virtually any element based on almost any property of the system. Controls are included in a scenario when they are specified in the Operational Alternative. The controls become part of an Operational Alternative when you specify the name of a Control Set to use in a given Operational Alternative.

The Control Manager is the main work center for controls. The Control Manager manages all controls, conditions, actions, and control sets in the system. The Control manager allows you to define controls using advanced IF, AND, and OR condition logic, which can trigger any number of THEN or optional ELSE actions.

Choose Components > Controls to open the Control Manager.



The Control Manager consists of the following tabs:

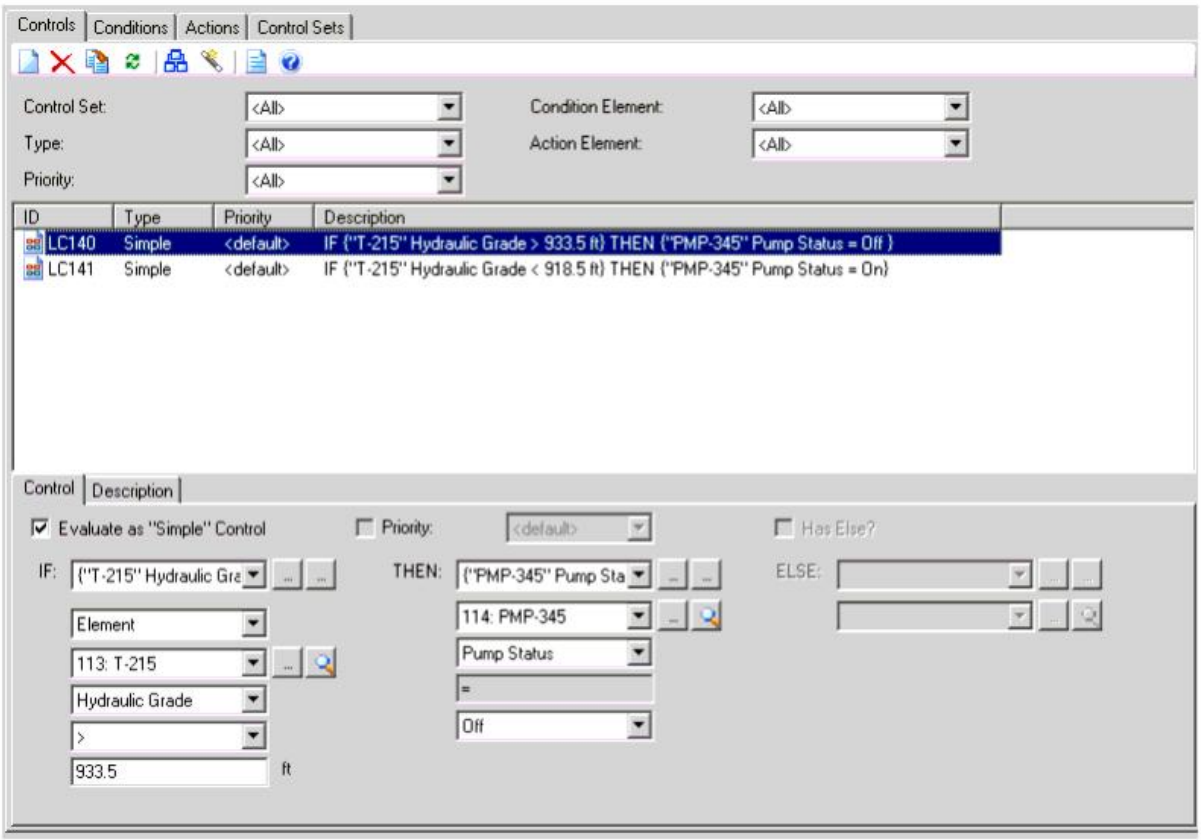
- **Controls** – Manage all controls defined in the system.
- **Conditions** – Define the condition that must be met prior to taking an action.
- **Actions** – Define what should be done to an element in the system in response to an associated control condition.
- **Control Sets** – Assign groups of controls to Control Sets.

Controls Tab

The Controls tab allows you to manage all controls defined in the system. Controls can be one of two types: simple or logical. Simple controls are made up of an IF condition and a THEN action statement. Logical controls are made up of an IF condition, a THEN action, and an optional ELSE action, and can be assigned a priority for resolving potential conflicts between logical controls.

Controls, Conditions, and Actions are assigned a non-editable application-provided ID (e.g., LC01).

The Controls tab is divided into sections:



- The pane in the center of the dialog box is the Controls List. This list displays a list of all Logical Controls defined in the system.
- Located above the Controls List is a toolbar with the following buttons:
  - **New**—Creates a new control.
  - **Delete**—Deletes the highlighted control. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.
  - **Duplicate**—Creates a copy of the currently highlighted control.
  - **Refresh**—Refreshes the highlighted control
  - **Control Sets**—Opens the Control Set dialog.
  - **Control Wizard**—Opens the Control Wizard.
  - **Report**—Generates a summary of the selected control, listing the ID, conditions, actions, and elements incorporated into the control.
- Below the toolbar is a set of filters that allow you to only display controls that meet criteria defined by the filter settings. The following filters are available:
  - **Type**—When a Type filter other than <All> is specified, only controls of that type will be displayed in the Controls list.
  - **Priority**—When a Priority filter other than <All> is specified, only controls of that priority will be displayed in the Controls list.
  - **Condition Element**—When a Condition filter other than <All> is specified, only controls containing the selected Condition element will be displayed in the Controls list.
  - **Action Element**—When an Action filter other than <All> is specified, only controls containing the selected Action element will be displayed in the Controls list.

You can edit or create controls consisting of an IF condition, a THEN action, and an optional ELSE action. The lower pane is split into sections:

- **Evaluate as Simple Control**—Turn on in order to evaluate the condition as a simple control.
    - **IF Condition**—The drop-down list allows you to choose from a list of conditions that have already been created in the Conditions tab.
    - **THEN Action**—The drop-down list allows you to choose from a list of actions that have already been created in the Actions tab.
    - **ELSE Action (optional)**—The ELSE action is used when the conditions for the control are not met. To specify an ELSE action, click the check box to activate the drop-down list. The drop-down list allows you to choose from a list of actions that have already been created in the Actions tab.
- **Priority**—This area of the dialog box is optional. To set a priority for the control being created, turn on to activate the priority drop-down list. You can set a priority of 1-5, 5 being the highest priority. If multiple controls meet a certain condition and they have conflicting actions, the control with the highest priority will be used.

    Note: At calculation time, the priority is used to determine the logical control to apply when multiple controls require that conflicting actions be taken. Logical controls with identical priorities will be prioritized based on the order they appear in the Logical Control Set alternative. A rule without a priority value always has a lower priority than one with a value. For two rules with the same priority value, the rule that appears first is given the higher priority.

    Relative speed pump patterns take precedence over any controls (simple or logical) that are associated with the pump.

    Hovering the mouse cursor over a control in the list will open a tooltip which displays the conditions and actions that make up that control.

    When creating a new condition or action for a new control, the condition and action input fields will be initialized with the data used in the last condition or action that was created.

    Once created, the Logical Control will be assigned an application generated ID (e.g., LC04).

- **Description**—This area is preset with a default description. There is an option to change the default description. To do so, turn on to activate the description field, and enter your description in the text box.
- **Summary**—This area of the dialog box displays a description of the control.
- **Status Pane**—When one or more filters are active, the lower left corner of the dialog will show the number of controls currently displayed out of the number of total controls. Additionally, a FILTERED flag is displayed in the lower right corner.

Logical, or rule-based controls allow far more flexibility and control over the behavior of your network elements than is possible with simple controls. This is accomplished by allowing you to specify one or more conditions and then link these to one or more Actions by using logical IF, AND, THEN, OR, and ELSE statements.

Note: Logical Controls are not executed during Steady State analyses.

Logical controls consist of any combination of simple conditions and simple actions. Controls are defined as:

IF

Condition 1 AND condition 2 OR condition 3 AND condition 4, etc., where condition X is a condition clause.

THEN

Action 1 AND action 2, etc. where action X is an action clause.

ELSE (Optional)

Action 3 AND action 4, etc. where action X is an action clause.

Priority (Optional)

Priority where priority is a priority value (1 to 5, 5 being the highest priority).

In addition to the high level of flexibility provided by allowing multiple conditions and actions, the functionality of Logical controls is also enhanced by the range of Condition types that are available. You can activate the stated actions based on element demands, element hydraulic grade or pressure, system demand, clock time, time from start, tank level, or time to fill or drain a tank.

You can also create composite conditions and actions. You can cause actions to be performed when multiple conditions are met simultaneously, or when one or the other conditions are met. You can also activate multiple actions when a single condition is met.

To create a logical control in which a pump (PMP-1) is turned on when the level in a tank (T-1) falls below a specified value (5 ft.) or when the system demands exceed a certain level (5000 gpm):

This example illustrates the power of using logical controls. To achieve the same functionality using simple controls, you would need to create four separate controls—one to turn the pump on if the tank level is below the specified value, one to turn the pump off if the tank level is above a specified value, one to turn the pump on if the system demand is greater than the specified value, and one to turn the pump off if the system demand is less than the specified value.

Note: Use the optional ELSE field to cause actions to be performed when the conditions in the control are not being met. For example, if you are creating a control that states, "If the level in Tank 1 is less than 5 ft., Then turn Pump 1 On," use an ELSE action to turn the pump off if the tank level is above 5 ft.

Note: Logical Controls are not executed during Steady State analyses.

When defining a logical control, you have the option to share conditions and/or actions. In other words, more than one control can reference the same condition or action. Keep in mind that when you change an underlying condition or action, it will affect all controls that reference that condition or action.

10.2 Calculation Options

Calculations depend on a variety of parameters that may be configured by you.

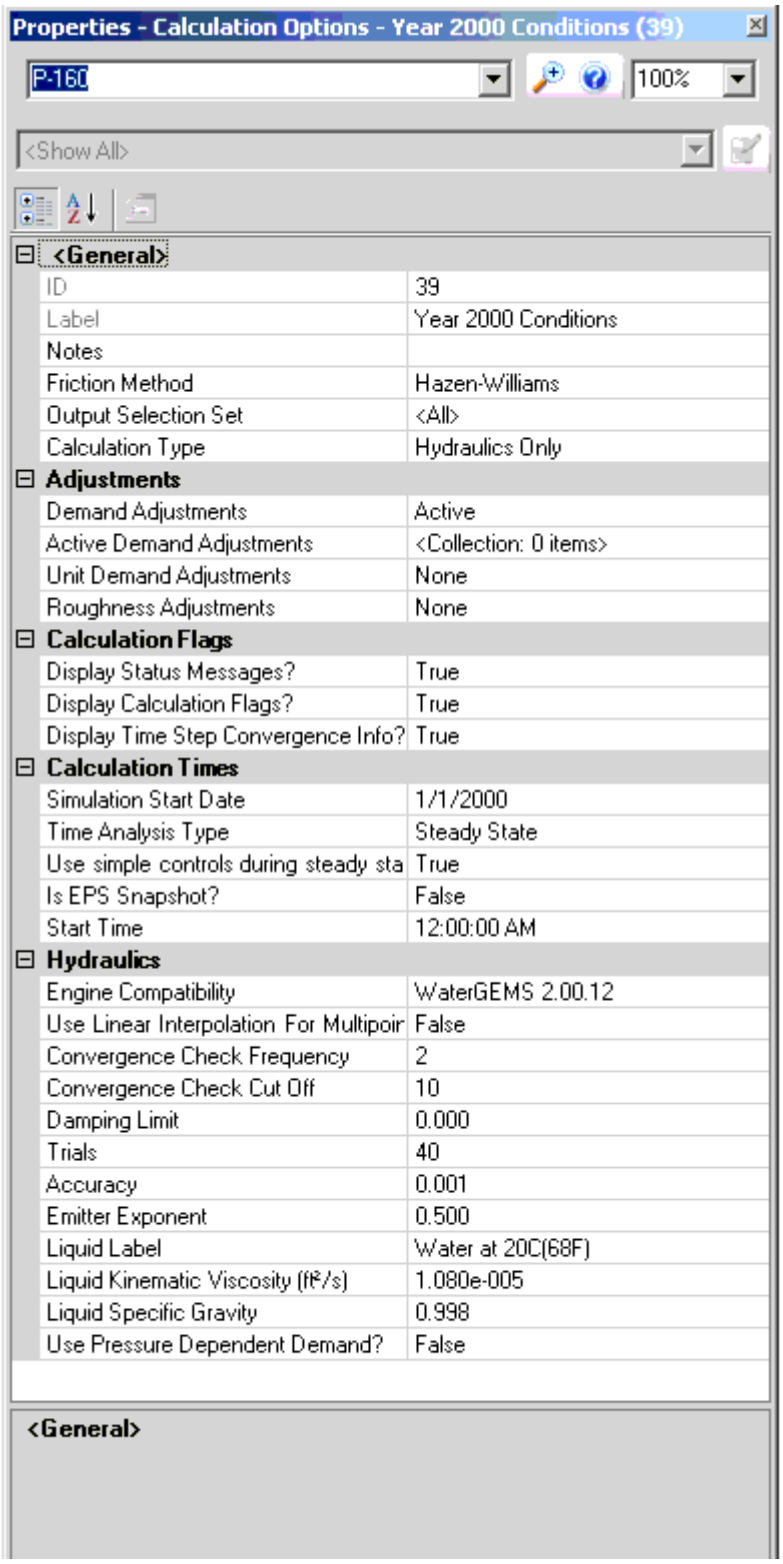
Choose Analysis > Calculation Options, Alt+3, or click the

New	Creates a new calculation option.
Duplicate	Makes a copy of the selected calculation option.
Delete	Deletes the selected calculation option. The base calculation option cannot be deleted.
Rename	Renames the selected calculation option.
Help	Displays online help for the Calculation Options.

The following controls are available from the Calculation Options dialog box.

To view the Steady State/EPS Solver properties of the Base Calculation Options

Select Base Calculation Options under Steady State/EPS Solver and double click to open the Properties dialog box.



The following calculation option parameters are available for user configuration:

- **Friction Method** – Set the global friction method.
- **Output Selection Set** –Select whether to generate output for All Elements (the default setting) or only the elements contained within the chosen selection set.
- **Calculation Type** – Select the type of analysis to perform with this calculation options set.
- **Consider Pumps and Valves in Min. System Pressure Constraints?** – If True the pressures at pumps and valves will be considered.
- **Demand Adjustments** – Specify whether or not to apply adjustment factors to standard demands.
- **Active Demand Adjustments** – The collection of demand adjustments that are applied during the analysis.
- **Unit Demand Adjustments** – Specify whether or not to apply adjustment factors to unit demands.
- **Active Unit Demand Adjustments** – The collection of unit demand adjustments that are applied during the analysis.
- **Roughness Adjustments** – Specify whether or not to apply adjustment factors to roughnesses.
- **Active Roughness Adjustments** – The collection of roughness adjustments that are applied during the analysis.
- **Display Status Messages?** – If set to true, element status messages will be stored in the output and reported.
- **Display Calculation Flags?** – If set to true, calculation flags will be stored in the output and reported.
- **Display Time Step Convergence Info?** – If set to true, convergence/iteration data for each time step will be stored in the output file and displayed in the calculation summary.
- **Simulation Start Date** – Select the calendar date on which the simulation begins.
- **Time Analysis Type** – Select whether the analysis is extended period or steady-state.
- **Use simple controls during steady state?** – When True, simple controls will be active during steady state analyses, else they will not be used. Note that logical controls are never used during steady state analysis.
- **Is EPS Snapshot?** – If True then an EPS snapshot is run instead of a regular steady state run. An EPS snapshot is a steady state run, but it considers the starting date and time of analysis and applies the appropriate pattern multipliers for that time. Note that since an EPS is not run, attributes such as tank levels are derived from the same initial conditions as a steady state run.
- **Equivalent Hydraulic Time Step** – In order that the pattern multipliers used in an EPS snapshot run exactly match those in an equivalent EPS run, specify the hydraulic time step of the EPS run that you wish to match.
- **Start Time** – Select the clock time at which the simulation begins.
- **Duration** – Specify the total duration of an extended period simulation.
- **Hydraulic Time Step** – Select the length of the calculation time step.
- **Override Reporting Time Step?** – Specify if you want the Reporting Time Step to differ from the Hydraulic Time Step.
- **Reporting Time Step** – Data will be presented at every reporting time step. The reporting time step should be a multiple of the hydraulic time step.

- **Set Water Quality Time Step?** – If set to True the Water Quality Time Step can be adjusted, otherwise it is computed by the calculation engine. It is not recommended that you set this to True.
- **Water Quality Time Step** – Time interval used to track water quality changes throughout the network. By default, this value is computed by the numerical engine and is equivalent to the smallest travel time through any pipe in the system.
- **Engine Compatibility**– This field allows you to choose which engine compatibility mode you want to run in. Choose WaterGEMS 2.00.12 to get all of the latest engine improvements and fixes made by Bentley and an engine mode that is based upon EPANET 2.00.12. This is the default setting for new models. Choose WaterGEMS 2.00.10 to maintain compatibility with previous version of WaterGEMS (V8i SELECTseries 1 and earlier), where the computational engine is based on EPANET 2.00.10. This is the default for upgraded models. If you select one of the EPANET modes, any enhancements, calculation corrections, and bug fixes made by Bentley will be disabled in order to match EPANET version results. Imported EPANET models will default to the appropriate EPANET version.
- **Use Linear Interpolation for Multipoint Pumps?** – If set to true the engine will use linear interpolation to interpret the pump curve as opposed to quadratic interpolation.
- **Convergence Check Frequency** – This option sets the number of solution trials that pass during hydraulic balancing before the status of pumps, check valves, flow control valves, and pipes connected to tanks are updated. The default value is 2, meaning that status checks are made every other trial. A value equal to the maximum number of trials would mean that status checks are made only after the system has converged (whenever a status change occurs the trials must continue since the current solution may not be balanced). The frequency of status checks on pressure reducing and pressure sustaining valves is determined by the Damping Factor calculation option.
- **Convergence Check Cut Off** – This option is the number of solution trials after which periodic status checks on pumps, check valves, flow control valves, and pipes connected to tanks are discontinued. Instead, a status check is made only after convergence is achieved. The default value is 10, meaning that after 10 trials, instead of checking status at every trial indicated by the Convergence Check Frequency setting, status is checked only at convergence.
- **Damping Limit** – This is the accuracy value at which solution damping and status checks on PRVs and PSVs should begin. Damping limits all flow changes to 60 percent of what would otherwise be as future trials unfold. The default of 0 indicates that no damping should be used and that status checks on control valves are made at every iteration. Damping might be needed on networks that have trouble converging, in which case a limit of 0.01 is suggested (relative to the default calculation hydraulic accuracy of 0.001).
- **Trials** – Unit-less number that defines the maximum number of iterations to be performed for each hydraulic solution. The default value is 40.
- **Accuracy** – Unit-less number that defines the convergence criteria for the iterative solution of the network hydraulic equations. When the sum of the absolute flow changes between successive iterations in all links is divided by the sum of the absolute flows in all links and is less than the Accuracy, the solution is said to have converged. The default value is 0.001 and the minimum allowed value for Accuracy is 1.0e-5.
- **Emitter Exponent** – Emitters are devices associated with junctions that model the flow through a nozzle or orifice. In these situations, the demand (i.e., the flow rate through the emitter) varies in proportion to the pressure at

the junction raised to some power. The constant of proportionality is termed the discharge coefficient. For nozzles and sprinkler heads the exponent on pressure is 0.5 and the manufacturer usually states the value of the discharge coefficient as the flow rate in gpm through the device at a 1 psi pressure drop.

- **Liquid Label** – Label that describes the type of liquid used in the simulation.
- **Liquid Kinematic Viscosity** – Ratio of the liquid's dynamic or absolute viscosity to its mass density.
- **Liquid Specific Gravity** – Ratio of the specific weight of the liquid to the specific weight of water at 4 degrees C or 39 degrees F.
- **Use Pressure Dependent Demand?** – If set to true the flows at junctions and hydrants will be based on pressure constraints.
- **Age Tolerance** – If the difference between two parcels of water is equal to or less than the value specified in this field, the parcels are considered to be of equal age.
- **Constituent Tolerance** – If the difference between two parcels of water is equal to or less than the value specified in this field, the parcels are considered to possess an equal concentration of the associated constituent.
- **Trace Tolerance** – If the difference between two parcels of water is equal to or less than the value specified in this field, the parcels are considered to be within the same percentile.

10.2.1 Fireflow analysis

One of the goals of a water distribution system is to provide adequate capacity to fight fires. Bentley WaterGEMS V8*i* 's powerful fire flow analysis capabilities can be used to determine if the system can meet the fire flow demands while maintaining various pressure constraints. Fire flows can be computed for a single node, a group of selected nodes, or all nodes in the system. A complete fire flow analysis can comprise hundreds or thousands of individual flow solutions – one for each junction selected for the fire flow analysis.

Fire flows are computed at user-specified locations by iteratively assigning demands and computing system pressures. The program calculates a steady-state analysis for each node in the Fire Flow Alternative. At each node, it begins by running a Steady-State analysis to ensure that the fire flow constraints that have been set can be met without withdrawing Fire Flow from any of the nodes. If the constraints are met in this initial run, the program then begins iteratively assigning the Needed Fire Flow demands at each of the nodes, and checking to ensure that the constraints are met.

The program then runs another set of Steady State analyses, this time either adding the Maximum Fire Flow (as set in the Fire Flow Upper Limit input box of the Fire Flow Alternative) to whatever normal demands are required at that node, or replacing the normal demands. In either case, the program checks the residual pressure at that node, the Minimum Zone Pressure, and, if applicable, the Minimum System Pressure. If the Fire Flow Upper Limit can be delivered while maintaining the various pressure constraints, that node will satisfy the Fire Flow constraints. If one or more of the pressure constraints is not met while attempting to withdraw the Fire Flow Upper Limit, the program will iteratively assign lesser demands until it finds the maximum flow that can be provided while maintaining the pressure constraints. If a node is not providing the Fire Flow Upper Limit, it is because the Residual Pressure at that node, the Minimum Zone Pressure, or the Minimum System Pressure constraints are not met while attempting to withdraw the Fire Flow Upper Limit (or the maximum number of iterations has been reached). If a node completely fails to meet the Fire Flow constraints, it is because the network is unable to deliver the Needed Fire Flow while still meeting the pressure constraints.

After the program has gone through the above process for each node in the Fire Flow Analysis, it runs a final Steady-State calculation that does not apply Fire Flow demands to any of the junctions. This provides a baseline of calculated results that can then be compared to the Fire Flow conditions, which can be determined by viewing the results presented on the Fire Flow tab of the individual junction editors, or in the Fire Flow Tabular Report. The baseline pressures are the pressures that are modeled under the standard steady-state demand conditions in which fire flows are not exerted.

Note: All parameters defining a fire flow analysis, such as the residual pressure or the minimum zone pressure, are explained in detail in the Fire Flow Alternative (see Fire Flow Alternative) and in the Fire Flow tab topics.

An online Tutorial on Fire Flow can be found by selecting the Help > Tutorials menu.

To perform a Fire Flow analysis

1. Change the **Calculation Type** calculation option to **Fire Flow (see** Calculation Options).
2. Open the **Alternatives** manager (Click the Analysis menu and select Alternatives).
3. Double click on **Base-Fire Flow** to open the Fire Flow Alternative editor.
4. Define the needed fireflow, fireflow upper limit, pressure constraints and the fire flow nodes selection set.
5. After all necessary fields have been entered, close the Fire Flow Alternative and Alternatives manager and click Compute . 
6. Open the Fire Flow Results Browser. Only the  elements that were specified in the selection set will be color coded.

11 MODEL CALIBRATION

11.1 What is Model Calibration?

- Model calibration is the process of comparing the model results to field observations, and, if necessary, adjusting the data in the model that describes the system until model-predicted performance reasonably agrees with measured system performance over a wide range of operating conditions.
- To calibrate the model under Steady State conditions the model pressures are matched to recorded pressures using measured flows. System operating parameters at the time the recorded pressures were taken are to be inputted into the model prior to calibration.

11.2 Why is Model Calibration Important?

- **Confidence:** Results provided by a computer model are frequently used to aid in making decisions regarding the operation or improvement of a hydraulic system. Calibration demonstrates the model's ability to reproduce existing conditions, thereby increasing the confidence the engineer will have in the model to predict system behavior.

- **Understanding:** The process of calibrating a hydraulic model provides excellent insight into the behavior and performance of the hydraulic system. In particular, it can show which input values the model is most sensitive to, so the model knows to be more careful in determining those values. With a better understanding of the system, the modeler will have an idea of the possible impact of various capital improvements or operational changes.

- **Troubleshooting:** One area of calibration that is often overlooked is the ability to uncover missing or incorrect data describing the system, such as incorrect pipe diameters, missing pipes, or closed valves. Thus, another benefit of calibration is that it will help in identifying errors caused by mistakes made during the model-building process.

11.3 What is the Normal Calibration Approach?

- Identify the intended use of the model to establish the level of detail needed in the model, the nature of the data collection, and the acceptable level of tolerance for errors between field measurements and simulation results.
- Determine estimate of model parameters.
- Collect calibration data.
- Evaluate model results based on initial estimates of model parameters.
- Perform a rough-tuning or macro-calibration analysis to correct larger discrepancies.
- Perform a sensitivity analysis, i.e. how performance of the calibration changes with respect to parameter adjustments, and then make a decision on the effort directions.
- Perform a fine-tuning or micro-calibration analysis to bring it to the desired level.

11.4 What should be Adjusted to Reach a Satisfactory Calibration?

- The process of calibration may include changing system demands, fine-tuning the roughness of pipes, altering pump operation characteristics, and adjusting other

model attributes that affect simulation results. Any and all input data that have uncertainty associated with them are candidates for adjustment during calibration to obtain reasonable agreement between model-predicted behavior and actual field behavior.

- **Pipe diameter & roughness coefficient:** the most common adjusted parameters. Pipe diameters can be checked for typographical error by sorting or color-coding pipes after the model is constructed. Usually, making adjustments to roughness coefficients is the preferable means of fine-tuning a model calibration. The reason is that considering diameter as an unknown would double the number of variables that must be determined for each pipe, and minimizing the number of variables is always desirable.

- **System demands:** In reality, water is withdrawn along the entire pipe, but in the model water usage within a certain area are grouped together and assigned to a single junction or distributed evenly to several junctions. This is to simplify the complexity of the modeling problem, which would cause discrepancies between the model results and field observations. Also, demand variation within a 24-hr period is another factor need to be considered during an extended period model calibration process.

- **Pipe length, node elevations, pipe/node connectivity, etc.** These data usually come from water system map. An outdated system map may have a pronounced effect on the accuracy of a model calibration effect.

- **Pump characteristic curves:** Normal wear and tear on a pump as it ages can cause the field performance to deviate from the performance illustrated on the pump curve supplied by the manufacturer. In this case, new curves should be determined and entered into the model based on field tests.

11.5 What are the Suggested Criteria for Model Calibration?

- See the table (“Criteria for Hydraulic Network Model Calibration”) on the next page.

11.6 What is Model Validation and How Important is it?

- Model validation is a further step after the model is calibrated, which involves comparisons between the model results and field observations using test data obtained under different conditions. Again, system demands, initial conditions and operational rules are adjusted to match the conditions at the time the test data was collected.
- Importance: The modeler can gain more confidence in the model and/or identify its shortcomings by validating it.
- While it is desirable to validate every model, most utilities do not have the time or money required to perform a thorough verification of the entire system. It is advised to perform a quick validation before applying the model calibrated years ago to a new problem.

Criteria for Hydraulic Network Model Calibration*

Intended Use	Level of Detail	Type of Simulations	Number of Pressure Readings	Accuracy of Pressure Readings	Number of Flow Readings	Accuracy of Flow Readings
Long-range Planning	Low	Steady-state or Extended Period	10% of Nodes	±5 psi for 100% of readings	1% of Pipes	±10%
Design	Moderate to High	Steady-state or Extended Period	5% - 2% of Nodes	±2 psi for 90% of readings	3% of Pipes	±5%
Operations	Low to High	Steady-state or Extended Period	10% - 2% of Nodes	±2 psi for 90% of readings	2% of Pipes	±5%

* *Data from "Calibration Guidelines for Water Distribution System Modeling" by American Water Works Association Engineering Computer Applications Committee, 1999. It has not been accepted as standards.*

Each application of a model is unique, and thus it is impossible to derive a single set of guidelines to evaluate calibration. The guideline presented below also give some numerical guidelines for calibration accuracy; however, they are in no way meant to be definitive. A range of values is given for most of the guidelines to reflect the differences among water systems and the needs of model users.

- **Master planning for larger systems (includes piping 24 in. and larger):** The model should accurately predict HGL to within 5-10 ft. during times of peak velocities and to the accuracy of the elevation and pressure data during normal demands. It should also reproduce tank water level fluctuations to within 3-6 ft. for EPS (Extended Period Simulation) runs and match treatment plant/well/pump station flows to within 10-20%.

- **Pipeline sizing:** The model should accurately predict HGL to within 5-10 ft. at the terminal point of the proposed pipe for fire flow conditions and to the accuracy of the elevation and pressure data during normal demands. If the new pipe impacts the operation of a water tank, the model should also reproduce fluctuation of the tank within 3-6 ft.

- **Fire flow analysis:** The model should accurately predict static and residual HGL to within 5-10 ft. at representative points in each pressure zone and neighborhood during fire flow conditions and to the accuracy of the elevation data during normal demands. If fire flow is near maximum fire flow such that storage tank sizing is important, the model should also predict tank water level fluctuation to within 3-6 ft.

- **Subdivision design:** The model should reproduce HGL to within 5-10 ft. at the tie-in point for the subdivision during fire flow tests, and to the accuracy of the elevation data during normal demands.

- **Distribution system rehabilitation study:** The model should reproduce static and residual HGL in the area being studied to within 5-10 ft. during fire hydrant flow tests, and to the accuracy of the elevation data during normal demands.

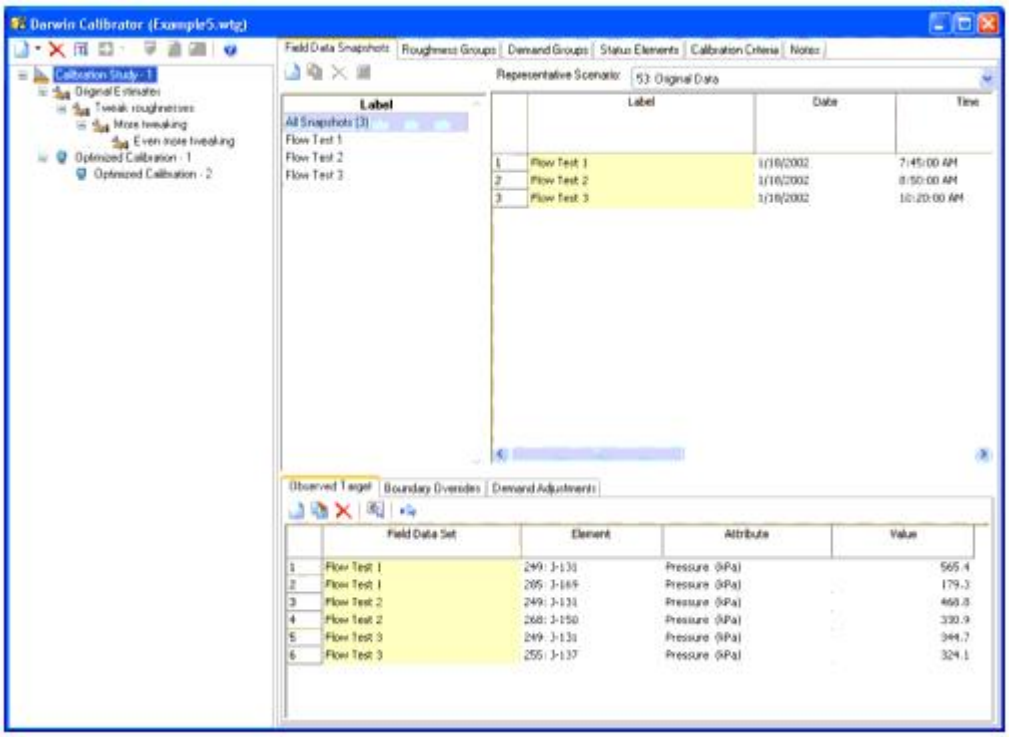
- **Energy use:** The model should reproduce total energy use over a 24-hour period to within 5-10%; reproduce energy consumption on an hourly basis to within 10-20%; and reproduce peak energy demand to within 5-10%.
- **Operational problems:** The model should reproduce problems occurring in the system such that the model can be used for decision-making for that particular problem.

- **Emergency planning:** The model should reproduce HGL to within 10-20 ft. during situations corresponding to emergencies (for example, fire flow, power outage, or pipe out of service).

11.7 Calibrating Your Model with Darwin Calibrator

The Bentley WaterGEMS V8*i* Darwin Calibrator provides a history of your calibration attempts, allows you to use a manual approach to calibration, supports multiple field data sets, brings the speed and efficiency of genetic algorithms to calibrating your water system, and presents several calibration candidates for you to consider, rather than just one solution. You can set up a series of Base Calibrations, which can have numerous Child Calibrations that inherit settings from their parent Base Calibrations.

Use Base and Child Calibrations to establish a history of your calibration trials to help you derive a list of optimized solutions for your water system. Inheritance is not persistent. If you change the Base Calibration, the change does not ripple down to the Child Calibrations.



You can adjust your model to better match the actual behavior of your water distribution system by using the Darwin Calibrator feature. It allows you to make manual adjustments on the model as well as adjustments using genetic algorithm optimization.

The left pane of the Darwin Calibrator dialog box displays a list of each calibration study in the current project, along with the manual and optimized runs and calculated solutions that make up each study.

The following controls can be found above the list pane:

New	Clicking the New button opens a submenu containing the following commands: • **New Calibration Study** - Creates a new calibration study. • **New Optimized Run** - Creates a new optimized run. Use this command if you want Bentley WaterGEMS V8*i* to efficiently process and evaluate numerous trial calibrations of your water system. You can set the optimized calibration to deliver several solutions for you to review. • **New Manual Run** - Creates a new manual run. Use this command if you want to test fitness by adjusting roughness, demand, or status manually. If you have specific solutions in mind, Manual Calibration might let you quickly narrow-down or refine the number and measure of adjustments before you use the genetic algorithm.
Delete	Deletes the calibration study, manual run, or optimized run that is currently highlighted in the list pane. Deleting a study will also delete all runs that are a part of that study. Deleting a run will also delete any child runs based on it.
Rename	Renames the calibration study, manual run, or optimized run that is currently highlighted in the list pane.
Compute	Opens a submenu containing the following commands: • **Compute**: Computes the optimized or manual run that is currently highlighted in the list pane. • **Hierarchy**: Computes the highlighted optimized or manual run as well all the optimized or manual runs branching from it hierarchically. • **Children**: Computes the highlighted optimized or manual run as well as all the calibration runs derived from it. • **Batch Run**: Opens the Batch Run dialog, allowing you to select multiple runs to compute together.
Export to Scenario	Opens the Export to Scenario dialog box, allowing you to export the solution that is currently highlighted in the list pane to a new or existing scenario, alternative, and/or set of alternatives.
Report	Opens the Report Viewer, which displays a detailed report of the solution that is currently highlighted in the list pane.
Graph	Opens the Correlation Graph dialog box, which displays a graph of the solution that is currently highlighted in the list pane.
Help	Opens the online help.

Note: Calibrator (as well as Designer and Skelebrator) are components that initialize their data when first used, so one needs to at least open the component for those database fields to be created in the current model.

As an example, if you are trying to use ModelBuilder to import calibration data but have never opened Calibrator in this particular model, you will not see the "Field Data Snapshot" model type in the dropdown list for Table Type. This is because that database type and its associated fields haven't been initialized yet. You would click on Analysis>Darwin Calibrator first in the main menu. Once this is done, the Field Data Snapshot and other Calibrator related fields are created, and those options will then appear in the ModelBuilder dialogs.

The right side of the dialog contains controls that are used to define settings and input data for Calibration Studies and their component Manual and Optimized Runs. The controls available on the right side of the dialog box will change depending on what is highlighted in the list pane:

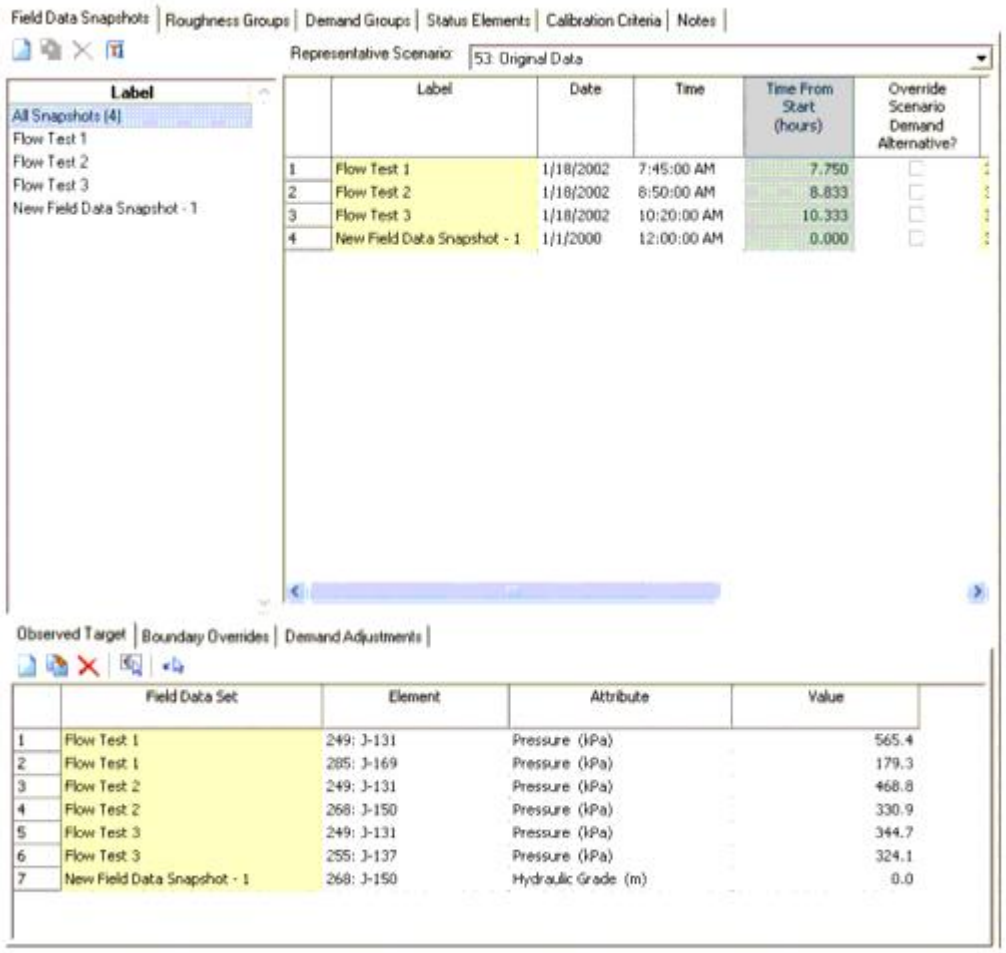
Calibration Studies
Optimized Runs
Manual Runs
Calibration Solutions

11.7.1 Calibration Studies

A Calibration Study is the starting point for all calibration operations. A Calibration study consists of the following components:

- Field Data Snapshots Tab
- Adjustment Groups
    - Roughness Groups
    - Demand Groups
    - Status Elements
- Calibration Criteria
- Notes (Optional).

Field Data Snapshots Tab
The Field Data Snapshots tab allows you to input observed field data for the calibration study that is currently highlighted in the list pane.



The following controls, located above the Field Data Snapshots list pane, allow you to manage your field data snapshots:

New	• Creates a new field data snapshot.
Duplicate	• Duplicates the currently highlighted field data snapshot.
Delete	• Deletes the currently highlighted field data snapshot.
Rename	• Renames the currently highlighted field data snapshot.

After a field data snapshot has been created, highlighting it in the list pane allows you to define or modify the following data:

Representative Scenario

Choose the scenario that will be used as the base data for the calibration study.

Snapshot Data

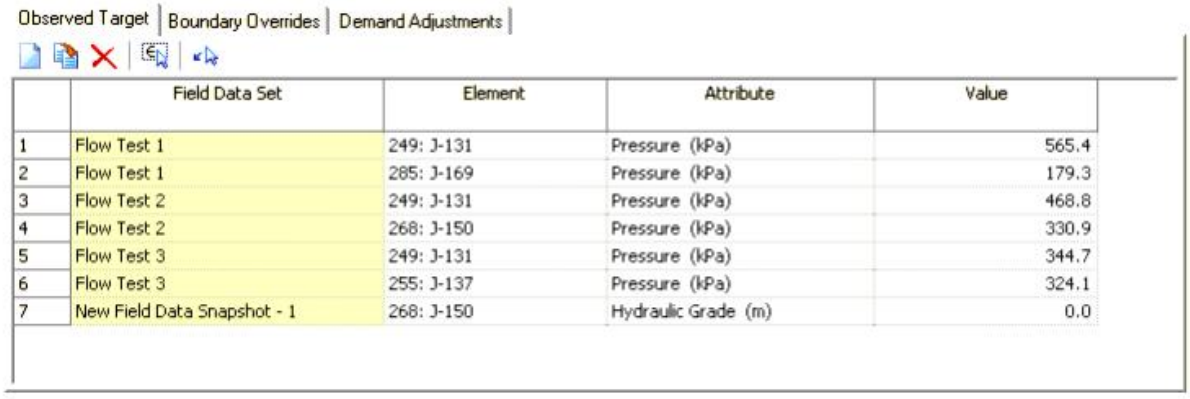
	Label	Date	Time	Time From Start (hours)	Override Scenario Demand Alternative?	Demand Alternative	Demand Multiplier	Notes
1	Flow Test 1	1/18/2002	7:45:00 AM	7.750	☐	30: 2002 Average Deman...	0.900	
2	Flow Test 2	1/18/2002	8:50:00 AM	8.833	☐	30: 2002 Average Deman...	0.900	
3	Flow Test 3	1/18/2002	10:20:00 AM	10.333	☐	30: 2002 Average Deman...	0.900	
4	New Field Data Snapshot - 1	1/1/2000	12:00:00 AM	0.000	☐	30: 2002 Average Deman...	0.000	

Enter the following Snapshot data:

Label	Enter a label for the field data snapshot.
Date	Set the date of the observations and field tests.
Time	Set the time of the observations and field tests. When using the pull down menu to select a time using the up and down arrows, hit the Enter key when you have selected the time you want to accept the change.
Time from Start	Displays the time difference from the time you set for the field data set to the time defined as the start of the scenario.
Override Scenario Demand Alternative?	Check this box to override the displayed Demand Alternative and use a different demand alternative or to use the specified Demand Multiplier. Clear this check box if you want to use the displayed alternative or if you do not want to use the Demand Multiplier.
Demand Alternative	Displays the Demand Alternative associated with the selected set of observations. If the Override Scenario Demand Alternative? box is checked, you can choose a different demand alternative here.
Demand Multiplier	Set a demand multiplier that is applied to your water model. For example, if you have knowledge that your demand is higher or lower by a specific percentage, you can set that value here. If the multiplier is set to zero, the demand will also be zero. By default this value is set to 1.
Notes	Use the Notes field to enter any comments you want saved with the field data snapshot.

Note: Field data set time is important since Calibrator uses the specified time to determine nodal demands from the representative scenario by applying pattern multipliers for the specified times. To that end be sure to specify the time that corresponds to the time the field data was acquired.

Observed Target



	Field Data Set	Element	Attribute	Value
1	Flow Test 1	249: J-131	Pressure (kPa)	565.4
2	Flow Test 1	285: J-169	Pressure (kPa)	179.3
3	Flow Test 2	249: J-131	Pressure (kPa)	468.8
4	Flow Test 2	268: J-150	Pressure (kPa)	330.9
5	Flow Test 3	249: J-131	Pressure (kPa)	344.7
6	Flow Test 3	255: J-137	Pressure (kPa)	324.1
7	New Field Data Snapshot - 1	268: J-150	Hydraulic Grade (m)	0.0

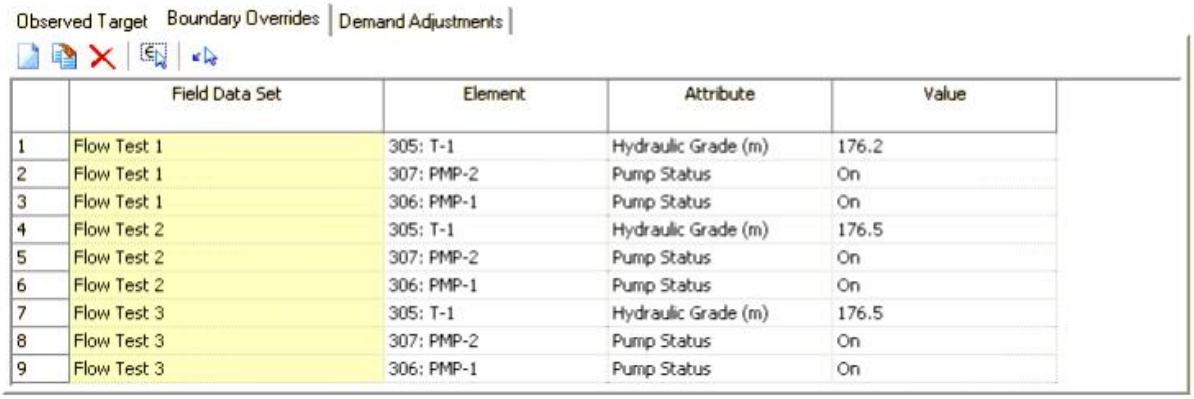
The Observed Target tab allows you to input calibration target values (node pressure and hydraulic grade line, as well as pipe flows) that the calibration operations will be attempting to match. Each row in the table represents a single target observation. The following controls are available in this tab:

New	Creates a new target observation for the Field Data Snapshot that is currently highlighted in the list.
Duplicate	Makes a copy of the currently highlighted target observation for the Field Data Snapshot that is currently highlighted in the list.
Delete	Deletes the currently highlighted target observation.
Initialize Table from Selection Set	Opens the Initialize From Selection set dialog, allowing you to choose a selection set. After a selection set is specified, this command generates a target observation for each element in the selection set.
Select From Drawing	Opens the Select dialog box, allowing you to select elements in the drawing view.

For each target observation, the table contains the following columns:

Field Data Set	Displays the field data set to which the target observation belongs.
Element	Select the element for which you want to enter observed data.
Attribute	Select the attribute for which you have observed data. Different attributes are available for each element type.
Value	Select a value from the drop-down list or enter in a value for the selected attribute.

Boundary Overrides



	Field Data Set	Element	Attribute	Value
1	Flow Test 1	305: T-1	Hydraulic Grade (m)	176.2
2	Flow Test 1	307: PMP-2	Pump Status	On
3	Flow Test 1	306: PMP-1	Pump Status	On
4	Flow Test 2	305: T-1	Hydraulic Grade (m)	176.5
5	Flow Test 2	307: PMP-2	Pump Status	On
6	Flow Test 2	306: PMP-1	Pump Status	On
7	Flow Test 3	305: T-1	Hydraulic Grade (m)	176.5
8	Flow Test 3	307: PMP-2	Pump Status	On
9	Flow Test 3	306: PMP-1	Pump Status	On

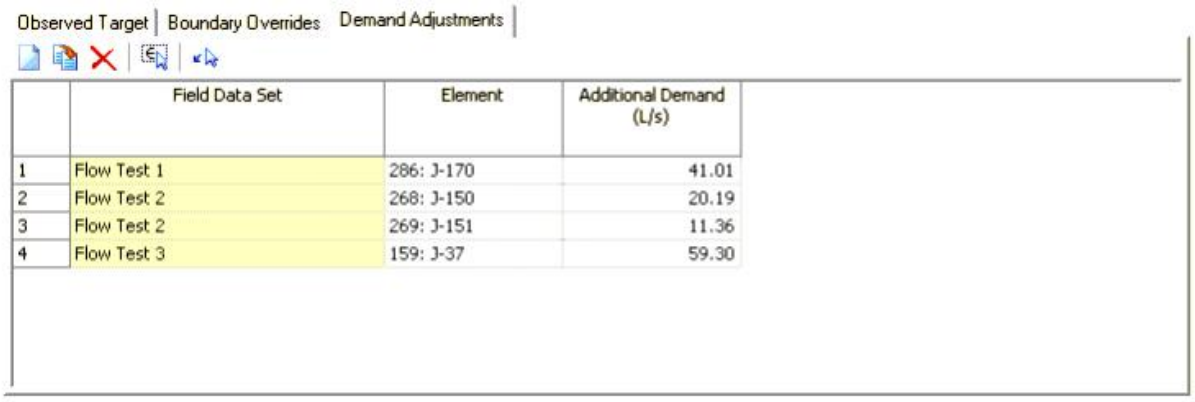
Observed boundary conditions such as tank level, pump status and speed and valve settings are entered in the Boundary Overrides tab. Each row in the table represents a single boundary override. The following controls are available in this tab:

New	Creates a new boundary override for the Field Data Snapshot that is currently highlighted in the list.
Duplicate	Makes a copy of the currently highlighted boundary override for the Field Data Snapshot that is currently highlighted in the list.
Delete	Deletes the currently highlighted boundary override.
Initialize Table from Selection Set	Opens the Initialize From Selection set dialog box, allowing you to choose a selection set. After a selection set is specified, this command generates a boundary override for each applicable element in the selection set.
Select From Drawing	Opens the Select dialog box, allowing you to select elements in the drawing view.

For each boundary observation, the table contains the following columns:

Field Data Set	Displays the field data set to which the boundary override belongs.
Element	Select the element for which you want to enter a boundary override.
Attribute	Select the attribute for which you have a boundary override. Different attributes are available for each element.
Value	Select a value from the drop-down list or type in a value for the selected attribute.

Demand Adjustments



	Field Data Set	Element	Additional Demand (L/s)
1	Flow Test 1	286: J-170	41.01
2	Flow Test 2	268: J-150	20.19
3	Flow Test 2	269: J-151	11.36
4	Flow Test 3	159: J-37	59.30

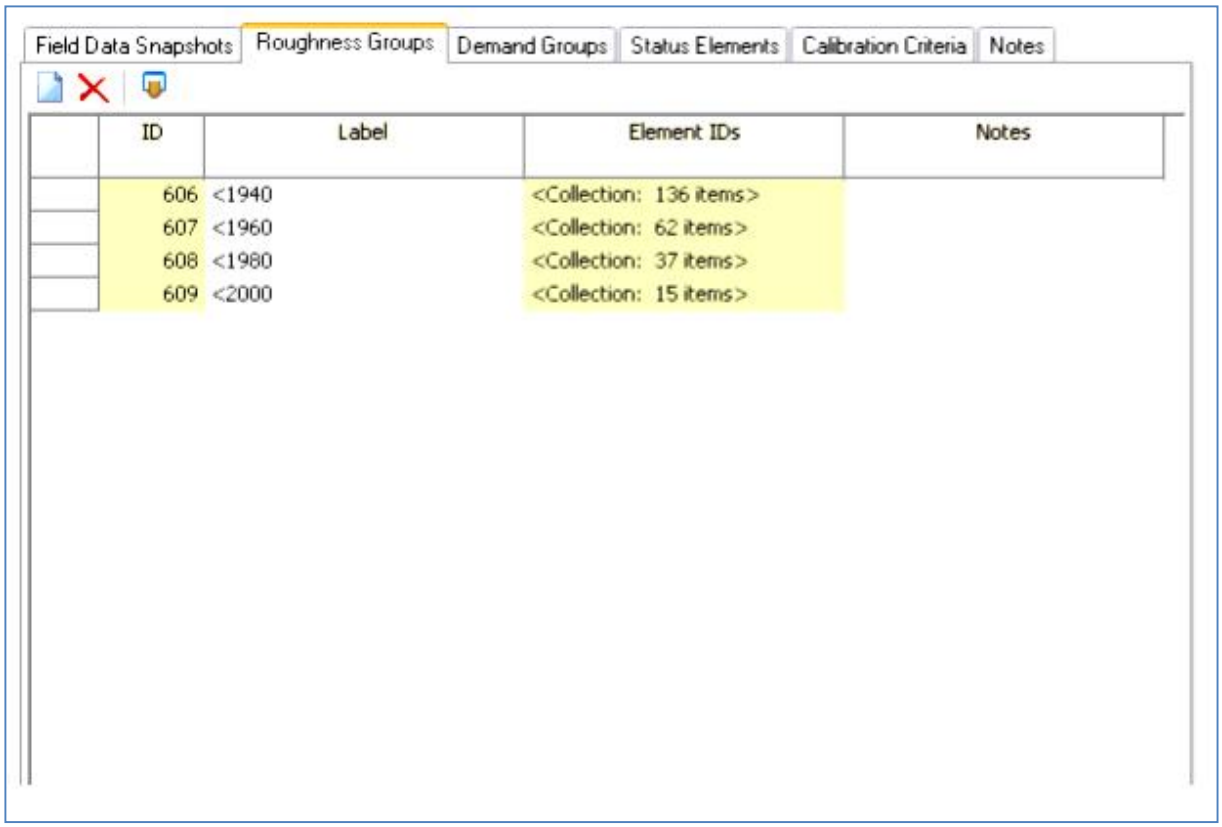
Use the Demand Adjustments tab to adjust demand for individual elements, such as flow from a hydrant. Additional demands (e.g., fire flow tests) are in addition to, not in lieu of, demands already calculated from pattern multipliers. Each row in the table represents a single demand adjustment. The following controls are available in this tab:

New	Creates a new demand adjustment for the Field Data Snapshot that is currently highlighted in the list.
Duplicate	Makes a copy of the currently highlighted demand adjustment for the Field Data Snapshot that is currently highlighted in the list.
Delete	Deletes the currently highlighted demand adjustment.
Initialize Table from Selection Set	Opens the Initialize From Selection set dialog, allowing you to choose a selection set. After a selection set is specified, this command generates a demand adjustment for each applicable element in the selection set.
Select From Drawing	Opens the Select dialog, allowing you to select elements in the drawing view.

For each demand adjustment, the table contains the following columns:

Field Data Set	Displays the field data set to which the demand adjustment belongs.
Element	Select the element for which you want to enter a demand adjustment.
Additional Demand	Type in a value for the demand adjustment.

Adjustment Groups
Adjustment groups are groups of elements whose attributes are adjusted together during the calibration process. You must be careful to group similar elements and not dissimilar ones. You can adjust the properties for a group as a whole but not for individual members of the group.



There are three kinds of adjustment groups, each of which are created and modified in their respective calibration study settings tab:

Roughness Groups - Add, edit, delete, or rename Roughness adjustment groups in the Roughness tab. Each roughness group should comprise elements that have similar attributes, such as pipes in a location of a similar material and age. Adjustments made to a group are applied to every element in the group. Click the Export Groups button to export the Calibration Group ID data to an automatically created user defined attribute. All elements within a calibration group will have an identical Calibration Group ID. This allows you to color code by calibration roughness group.

Demand Groups - Add, edit, delete, or rename Demand adjustment groups in the Demand tab. Adding Demand Calibration adjustment groups introduces more unknowns into a calibration problem. If available, you should enter more accurate demand data into your Bentley WaterGEMS V8*i* model, rather than adding Demand Adjustment Groups. Consider creating Demand Groups based on usage patterns. Click the Export Groups button to export the Calibration Group ID data to an automatically created user defined attribute. All elements within a calibration group will have an identical Calibration Group ID. This allows you to color code by calibration demand group.

You can automatically create demand groups from selection sets using the Group Generator. To open the Group Generator click the Create Multiple Design Groups button.

Status Elements - Add, edit, delete, or rename Status Element adjustment groups in the Status Elements tab. Status indicates whether a pipe is open or closed. GA-optimized calibration will identify the status of each pipe within the status group so that the chosen objective function is minimized. Status groups are generally used when a particular area of

the system is believed to contain a closed pipe or valve. We recommend that Status Groups comprise, at most only a few pipes, or one pipe. Click the Export Groups button to export the Calibration Group ID data to an automatically created user defined attribute. All elements within a calibration group will have an identical Calibration Group ID. This allows you to color code by calibration status group.

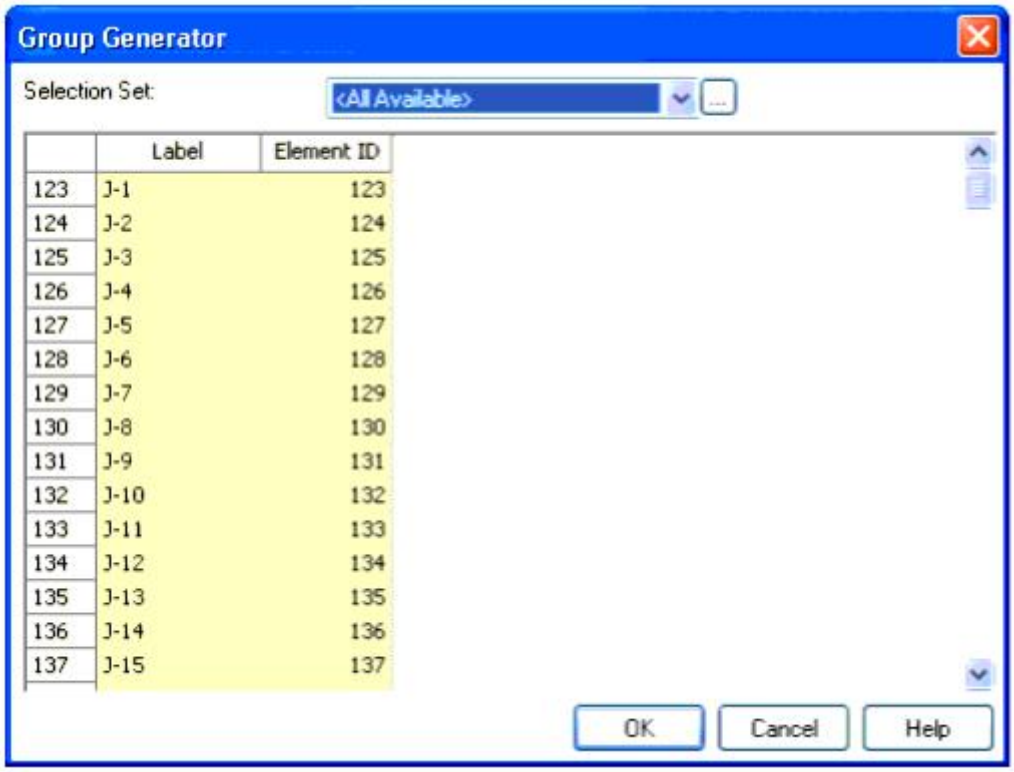
Each adjustment group tab consists of a table that lists the adjustment groups, a New button to add groups to the table, and a Delete button to remove the currently selected group from the table. The table consists of the following columns:

ID	The automatically assigned ID of the adjustment group.
Label	The user-defined name of the adjustment group. To change the label, click on it and type a new name.
Element IDs	The elements that are contained within the adjustment group. Clicking the ellipsis button in this field will open the Selection Set dialog, which allows you to add and remove elements by selecting them in the drawing view.
Notes	Use the Notes field to enter any comments you want saved with the adjustment group.

Note: Decide on your Adjustment Groups first and then collect the Field Data to support the number or groups, rather than letting available data determine how many Adjustment Groups you have.

Group Generator Dialog Box

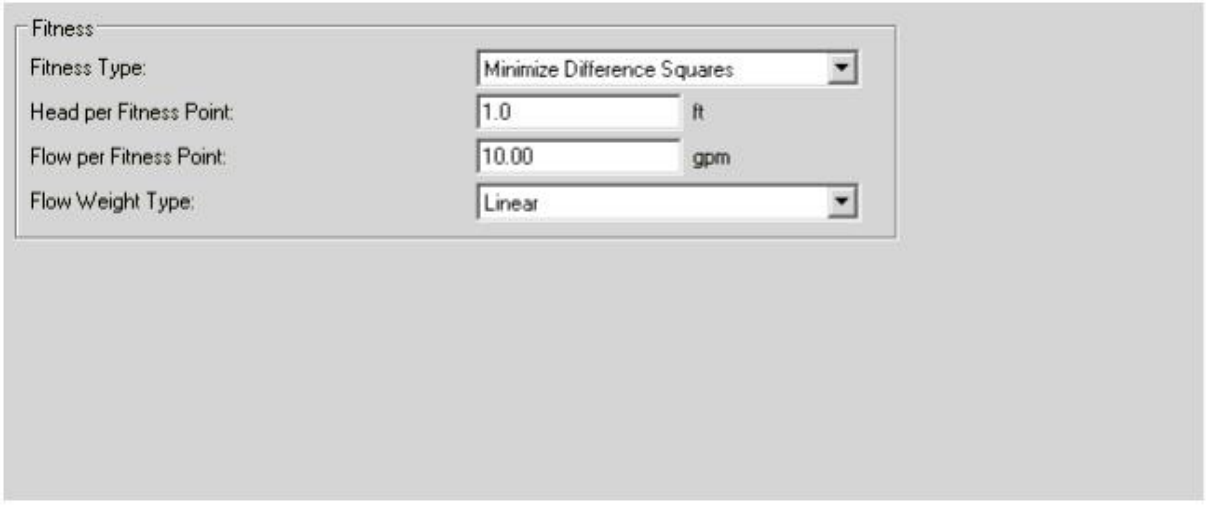
The Group Generator allows you to automatically create multiple design groups based on existing selection sets, or by selecting a group of elements from the drawing.



The dialog consists of a list of elements that will be used to create demand groups (one element per group) and a menu that allows you to select the elements that are included in the list. The menu contains a list of all existing selection sets. Click the elipsis button to select elements from the drawing directly. When the list contains all of the elements that you want to be included in demand groups, click OK.

Calibration Criteria

Use the Calibration Criteria tab to set up how the calibrations are evaluated.



The options you specify are applied to every calibration trial in the Calibration Study. The Calibration Criteria tab contains the following controls:

- **Fitness Type –** Select the Fitness Type you want to use from the drop down list. In general, regardless of the fitness type you select, a lower fitness indicates better calibration. Fitness Types include: Minimize Difference Squares, Minimize Difference Absolute Values, and Minimize Maximum Difference. For more information, see Calibration Criteria Formulae.
    - **Minimize Difference Squares -** Uses a calibration designed to minimize the sum of squares of the discrepancy between the observed data and the model simulated values. (Model simulated values include hydraulic grades and pipe discharges.) This calibration favors solutions that minimize the overall sum of the squares of discrepancies between observed and simulated data.
    - **Min. Diff. Absolute Values -** Uses a calibration designed to minimize the sum of absolute discrepancy between the observed data and the model simulated values. This calibration favors solutions that minimize the overall sum of discrepancies between observed and simulated data.
    - **Minimize Max. Difference -** Uses a calibration designed to minimize the maximum of all the discrepancies between the observed data and the model simulated values. This calibration favors solutions that minimize the worst single discrepancy between observed and simulated data. Note that the Minimize Maximum Difference Fitness Type is more sensitive to the accuracy of your data than other Fitness Types.
- **Head/Flow per Fitness Point -** Head and Flow per Fitness Type provide a way for you to weigh the importance of head and flow in your calibration. Set these values such that the head and flow have unit equivalence. You can give higher importance to Head or Flow by setting a smaller number for its Per Fitness Point Value.
- **Flow Weight Type -** Select the type of weight used: None, Linear, Square, Square Root, and Log. The weighting type you use can provide a greater or lesser fitness penalty.

  In general, measurements with larger flow carry more weight in the optimization calibrations than those with less flow. You can exaggerate or reduce the effect larger measurements have on your calibration by selecting different weight types. For example, using no weighting (None) provides no penalty for measurements with lesser flow versus those with greater flow. Using log and square root reduces the fitness penalty for measurements with lesser flow, and using linear or square increases the fitness penalty for measurements with less flow.

  Note: If you change the Calibration Options, any fitness values you get are not comparable to fitness values obtained using different Calibration Options settings.

12 MODEL APPLICATION

- **Master Planning:** This is the most common application of water distribution modeling. Models are used to identify existing system deficiencies, and predict what improvements and additions to the distribution system will be necessary to accommodate future customers.

- **Preliminary design:** In preliminary design, the engineer models the facilities that will be required to serve a particular area of, or addition to, the distribution system service area. Pipe and storage tank sizing, new pump selection can be realized by running the calibrated model.

- **Subdivision layout:** When designing a subdivision, fire flows usually dominate over customer demands, and the planning horizon is typically short. An added flow can be entered into the subdivision tie-in point to test the impact of this new demand on the remainder of the system.

- **Rehabilitation Analysis:** In a rehabilitation study of an area of a system, adequate capacity for fire flows is usually the most important consideration. Many more alternative scenarios are needed compared to designing new pipes, since a variety of possible solutions exist (for example, relining, paralleling, or looping). These alternatives can be tested and screened using a calibrated model.